PyroSim

A MODEL CONSTRUCTION TOOL FOR FIRE DYNAMICS SIMULATOR (FDS)

THUNDERHEAD ENGINEERING

1006 Poyntz Ave.
Manhattan, KS 66502-5459
785-770-8511
www.thunderheadeng.com

in collaboration with

RJA

The RJA Group, Inc.
One Pointe Drive, Suite 210
Brea, CA 92821-6315

PyroSim Example Guide

2007.2

Example Guide

Table of Contents

1. Before Starting 1
    Install PyroSim 1
    Units 1
    Manipulating the 3D Image 1
    FDS Concepts and Nomenclature 1
2. Example Problems Provided with FDS 5 3
    Ethanol Pan Fire 3
    Box Burn Away 4
    Insulated Steel Column 5
    Water Cooling 6
3. Burner Fire with Temperature Measurement 8
    Create the Mesh 8
    Create the Burner Surface 9
    Create the Burner Vent 10
    Create the Top Vent 11
    Rotate the Model for a Better View 12
    Add a Temperature Measurement Device 13
    Add a Temperature Slice Plane 13
    Save the Model 14
    Run the Simulation 14
    View Smoke in 3D 15
    View Temperature Slice Plane 15
    View Temperature Measurements 16
4. Smoke Layer Height and Heat Flow Through a Door 17
    Create the Burner Surface 17
    Create the Burner Vent 18
    Create the Open Side Vent 19
    Create the Mesh 20
    Add the Wall 21
    Add the Door 21
    Rotate the Model for a Better View 22
    Add a Layer Zoning Device 22
    Add a Flow Measuring Device 23
    Set the Simulation Length 23
    Save the model 23
    Run the Simulation 23
    View Smoke in 3D 24
    View Time History Data 25
5. Switchgear Fire Example 27
    Computational Mesh 29
    Material Properties 31
    Save the Model 33
    Surface Properties 33
    Model Geometry 35
    Post-Processing Controls 51
    Simulation Parameters 53

Run the Analysis 53
View the Results 54
References 58

List of Figures

2.1. Ethanol pan model 4
2.2. Ethanol pan results 4
2.3. Foam box burn away model 5
2.4. Foam box burn away results 5
2.5. Insulated Steel Column model 6
2.6. Insulated Steel Column results 6
2.7. Water cooling model 7
2.8. Water cooling results 7
3.1. Burner fire in this example 8
3.2. Creating the mesh 9
3.3. Creating a new burner surface 10
3.4. Defining parameters for the burner surface 10
3.5. Creating the burner vent 11
3.6. After creating the open vent 12
3.7. The model after rotating. The burner is shown in red and the top vent in blue. 13
3.8. The simulation dialog during the analysis. 14
3.9. The initial Smokeview display. 15
3.10. 3D smoke in the model. 15
3.11. Temperature contours on the slice plane. 16
3.12. Temperature time history plot. 16
4.1. Smoke in the model 17
4.2. Creating a new burner surface 18
4.3. Defining parameters for the burner surface 18
4.4. Creating the burner vent 19
4.5. Creating the mesh 20
4.6. Creating the wall 21
4.7. The model after rotating. The burner is shown in red and the top vent in blue. 22
4.8. The simulation dialog during the analysis. 24
4.9. The initial Smokeview display. 24
4.10. 3D smoke in the model. 25
4.11. Time history plot of heat flow through the door. 26
4.12. Time history plot of heat flow through the door. 26
5.1. Pictorial representation of the switchgear room complex, showing switchgear cabinets, cable trays, supply ducts and vents, and smoke detectors. Drawing not to scale. 27
5.2. Completed model 28
5.3. Input to create the mesh 30
5.4. Display of the mesh 31
5.5. Concrete properties 32
5.6. Composition of the concrete surface 34
5.7. The concrete surface 34
5.8. Input for the wall dividing the two rooms 36
5.9. Display of the dividing wall 37
5.10. Sketch of the lower left cabinet 40
5.11. Making a copy of Cabinet 1 by dragging the vertical handle. The final position will be 4 feet from the left and top boundaries. 41
5.12. The rooms showing the switchgear cabinets 42

5.13. The room showing the cables 45
5.14. Creating the supply vent surface 46
5.15. The room showing the vents 49
5.16. Defining the temperature isosurfaces 53
5.17. Completed model 54
5.18. Smoke in rooms at 105 seconds 55
5.19. Temperature contours on slice planes at 330 seconds 55
5.20. Temperature contours on the solid surfaces at 525 seconds 56
5.21. Layer height in room 3A 57

List of Tables

5.1. Room size (interior dimensions) 28
5.2. Door size 28
5.3. Concrete properties (NUREG/CR-1805) 28
5.4. Sheet metal properties (generic) 29
5.5. Cable properties (NUREG/CR-6850) 29
5.6. Dimensions for the mesh in Room 3A (including 2' thick walls) 30
5.7. Dimensions for the mesh in Room 3B (including 2' thick walls) 31
5.8. Dividing wall dimensions 36
5.9. Dividing door dimensions 37
5.10. Entrance door blockage dimensions 38
5.11. Cabinet Dimensions 40
5.12. Dimensions for Cabinet 2 41
5.13. Cable A Dimensions 43
5.14. Cable C Dimensions 44
5.15. Return vent dimensions 47
5.16. Supply duct dimensions 48
5.17. Supply vent dimensions 48
5.18. Return vent dimensions 48
5.19. Fire dimensions 51
5.20. Layer data for Room 3A 52
5.21. Layer data for Room 3B 52
5.22. Slice plane data 52

Chapter 1. Before Starting

Install PyroSim

In order to work through this tutorial you must be able to run PyroSim. You can download PyroSim from the internet by going to http://www.pyrosim.com/ to obtain the free trial.

Units

Except where noted, the instructions given in this tutorial will assume that PyroSim's current unit system is **SI**. If PyroSim is using a different unit system, the simulation will not produce the expected results. To ensure that you are using **SI** units:

1. In the **View** menu, click **Units**.
2. In the **Units** sub-menu, verify that **SI** is selected.

At any time, you can switch between SI and English units. The data is stored once in the original system, so there is no loss of accuracy when you switch units.

Manipulating the 3D Image

- To **spin** the 3D model, select then **left-click** on the model and move the mouse. The model will spin as though you have selected a point on a sphere.
- To **zoom**, select (or hold the **ALT** key) and drag the mouse vertically. Select then click and drag to define a zoom box.
- To **move** the model, select (or hold the **SHIFT** key) and drag to reposition the model in the window.
- To change the focus of the view, select an object(s) and then select to define a smaller viewing sphere around the selected objects. Selecting will reset the view to include the entire model.
- At any time, selecting (or pressing **CTRL + R**) will **reset** the model.

You can also use Smokeview and person-oriented controls. See the PyroSim User Manual for instructions.

FDS Concepts and Nomenclature

Material

Materials are used to define thermal properties and pyrolysis behavior.

Surface

Surfaces are used to define the properties of solid objects and vents in your FDS model. The surface can use previously defined materials in mixtures or layers. By default, all solid objects and vents are inert, with a temperature that is fixed at the initial temperature.

Obstruction

Obstructions are the fundamental geometric representation in FDS. Obstructions are rectangular solids defined by two points. Surface properties are assigned to each face of the obstruction. Activation events can be defined to create or remove an obstruction during a simulation.

The geometry of an obstruction does not need to match the geometry of the mesh used for the solution. However, the geometry can only be resolved to the mesh. At the time of analysis, all faces of an obstruction are shifted to correspond to the nearest mesh cell. Thus, some obstructions may become thicker in the analysis; others may become thin and correspond to a single cell face which has the potential to introduce unwanted gaps into a model. These ambiguities can be avoided by making all geometry correspond to the mesh spacing.

Vent

Vents have general usage in FDS to describe 2D planar objects. Taken literally, a vent can be used to model components of the ventilation system in a building, like a diffuser or a return. In these cases, the vent coordinates define a plane forming the boundary of the duct. No holes need to be created; air is supplied or exhausted by the vent.

You can also use vents as a means of applying a particular boundary condition to a rectangular patch on a surface. A fire, for example, can be created by specifying a vent on either a mesh boundary or solid surface. The vent surface defines the desired characteristics of fire.

Computational Mesh

FDS calculations are performed within a domain made of rectilinear volumes called *meshes*. Each mesh is divided into rectangular *cells*. Two factors that must be considered when choosing the cell size are the required resolution to define objects in the model (obstructions) and the desired resolution for the flow dynamics solution (including local fire induced effects). Although geometric objects (obstructions) in an FDS analysis can be specified using dimensions that do not fall on cell coordinates, during the FDS solution, all faces of an obstruction are shifted to the closest grid cell. If an obstruction is very thin, the two faces may be approximated on the same cell face. The FDS Users Guide [McGrattan, Klein, Hostikka, and Floyd, 2007] recommends that, for full functionality, obstructions should be specified to be at least one grid cell thick. As a result, the cell size must be selected small enough to reasonably represent the problem geometry. In addition, cells should be as close to cubes as possible.

Whether the cell size is sufficient to resolve the flow dynamics solution can only be determined by a **grid sensitivity study**. A discussion of model sensitivity to mesh size is given in Chapter 5 of Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications [U.S. NRC, 2007]. It is the responsibility of the analyst to perform a sensitivity study as part of any simulation.

Chapter 2. Example Problems Provided with FDS 5

If you want to have fun and quickly run some example analyses, you can import FDS 5 input files that are included with the NIST installation of FDS 5. These examples are provided with the PyroSim distribution in the PyroSim 2007\samples\FDS5 folder. In this chapter, we illustrate this using a few examples; you can import additional examples. Studying these examples is a good way to learn about input for different types of analyses.

To open any of these examples:

1. Obtain the desired FDS 5 input file.

2. Start PyroSim.

3. On the **File** menu, click **Import** and select **FDS File**.

4. In the **Open File** dialog, click the FDS input file. PyroSim will import the file. PyroSim will warn if any records on the FDS file cannot be imported.

5. On the **File** menu, click **Save**. Save the model in a new directory.

6. On the **FDS** menu, click **Run FDS...**. This will start the analysis. At the end of the analysis, SmokeView will be launched to view the results.

Important

If PyroSim doesn't fully support a FDS input file, it will issue a warning that includes information about how it handled the unfamiliar records. In some cases, PyroSim is able to add records to the **Additional Records** section and the simulation is unaffected. However, if records are "dropped" (i.e. omitted from the simulation) the simulation results will no longer represent the original intent of the example problem.

Ethanol Pan Fire

The **ethanol_pan.fds** example illustrates an ethanol pan fire. The model is shown in Figure 2.1. A typical result is shown in Figure 2.2

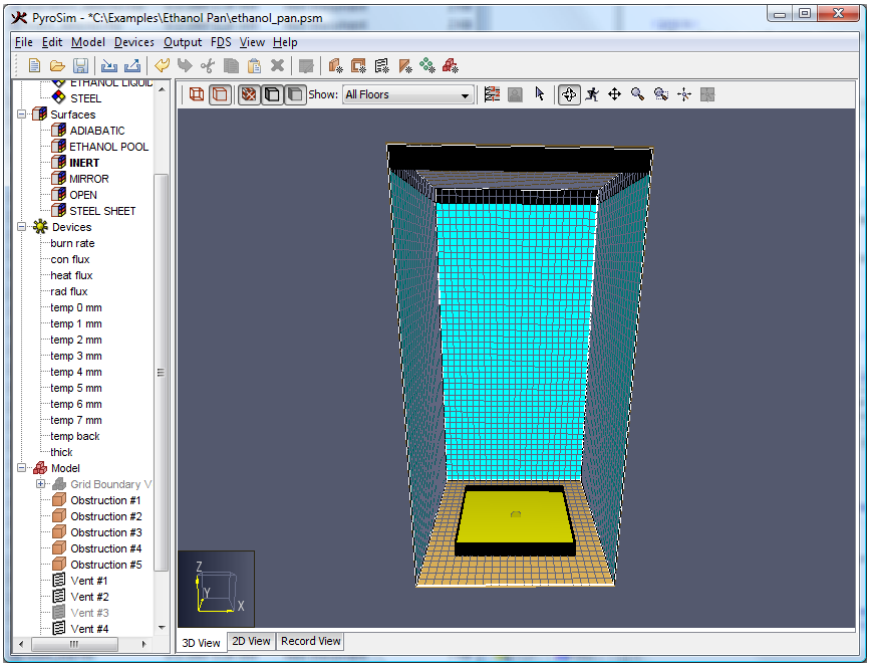

Figure 2.1. Ethanol pan model

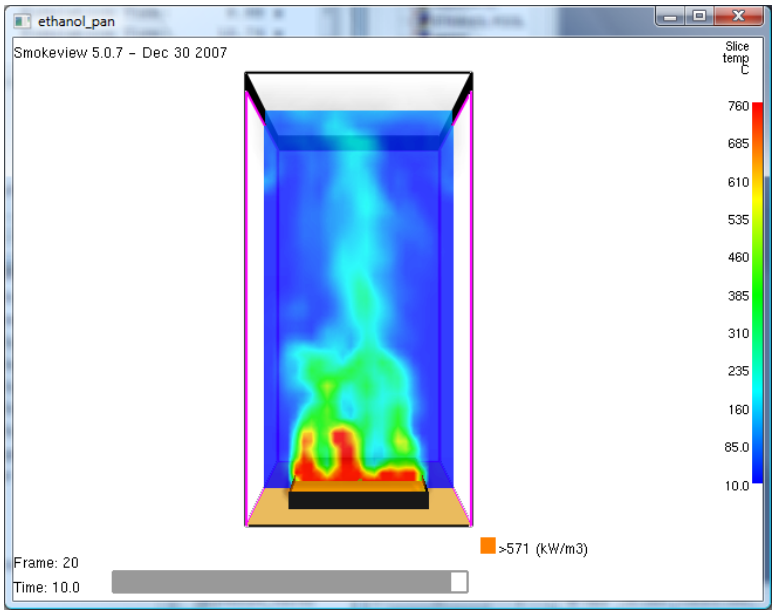

Figure 2.2. Ethanol pan results

Box Burn Away

The **box_burn_away.fds** example illustrates burning of a foam box. The model is shown in Figure 2.3. A typical result is shown in Figure 2.4

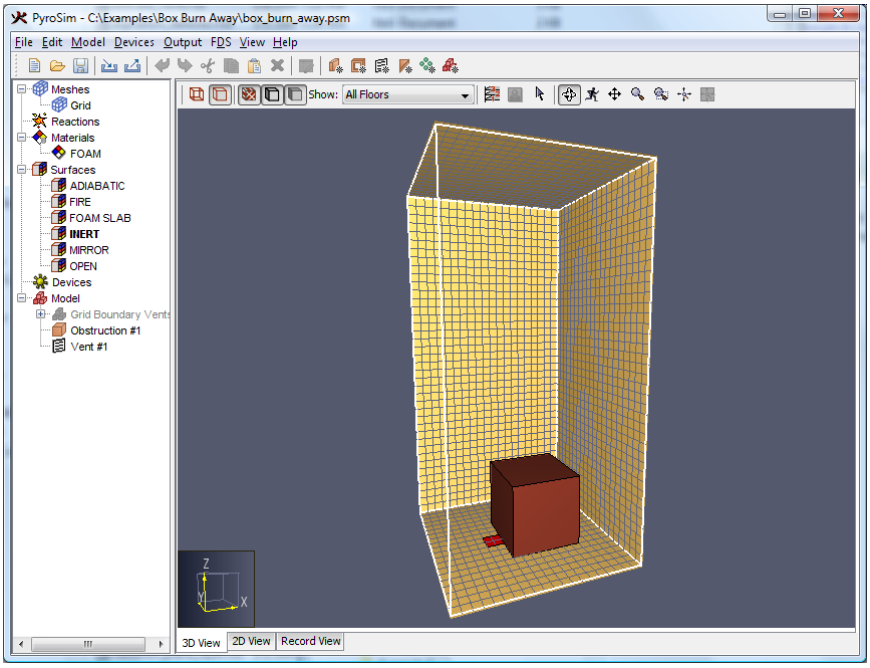

Figure 2.3. Foam box burn away model

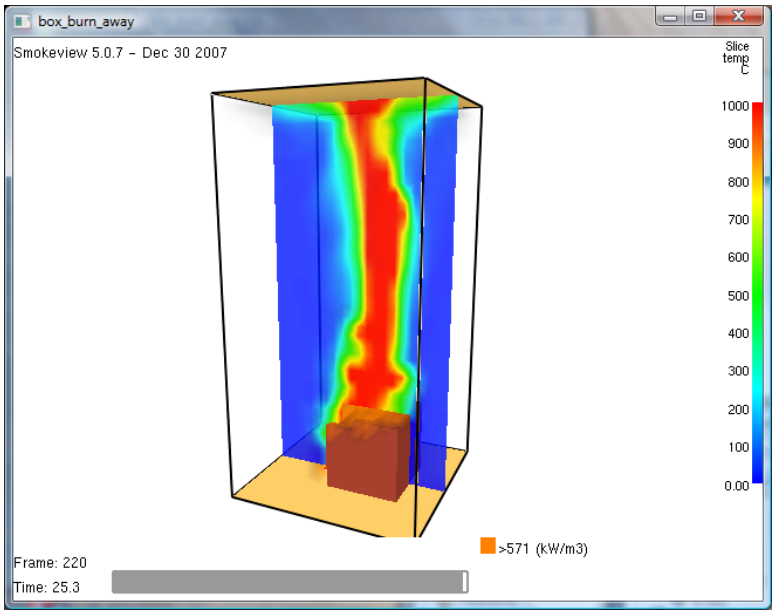

Figure 2.4. Foam box burn away results

Insulated Steel Column

The **insulated_steel_column.fds** example illustrates heat conduction into a column. The model is shown in Figure 2.5. A typical result is shown in Figure 2.6

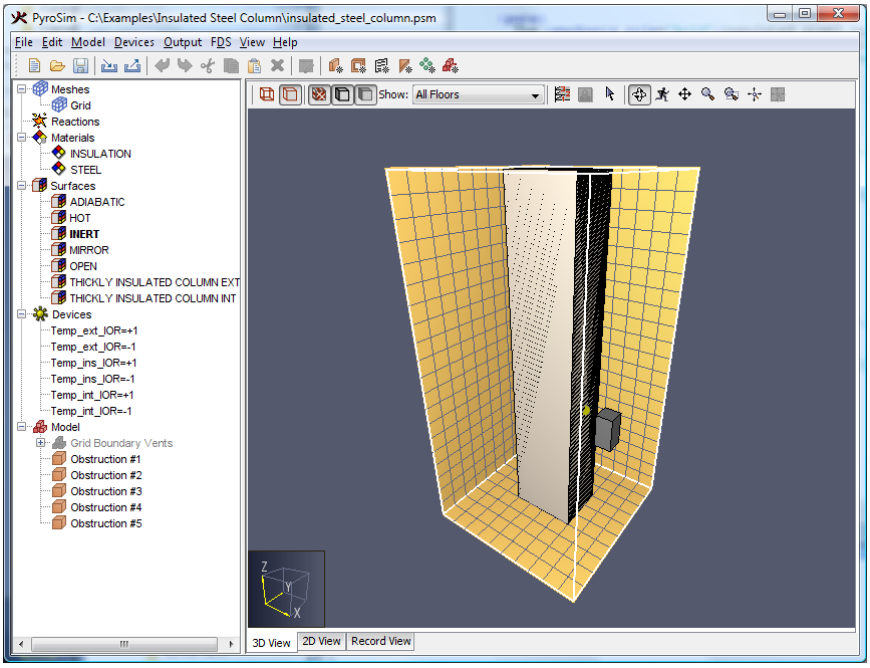

Figure 2.5. Insulated Steel Column model

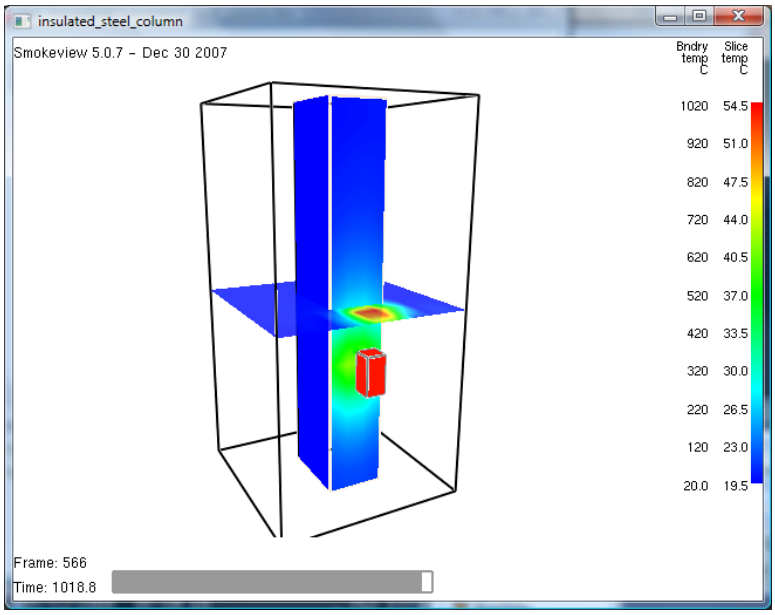

Figure 2.6. Insulated Steel Column results

Water Cooling

The **water_cooling.fds** example illustrates water cooling spray onto a wall. The model is shown in Figure 2.7. A typical result is shown in Figure 2.8

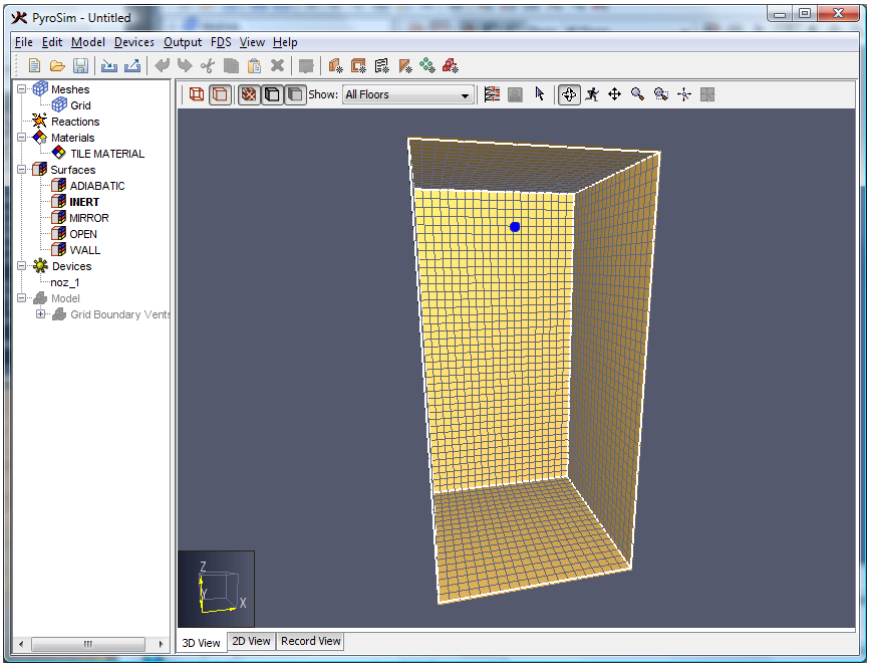

Figure 2.7. Water cooling model

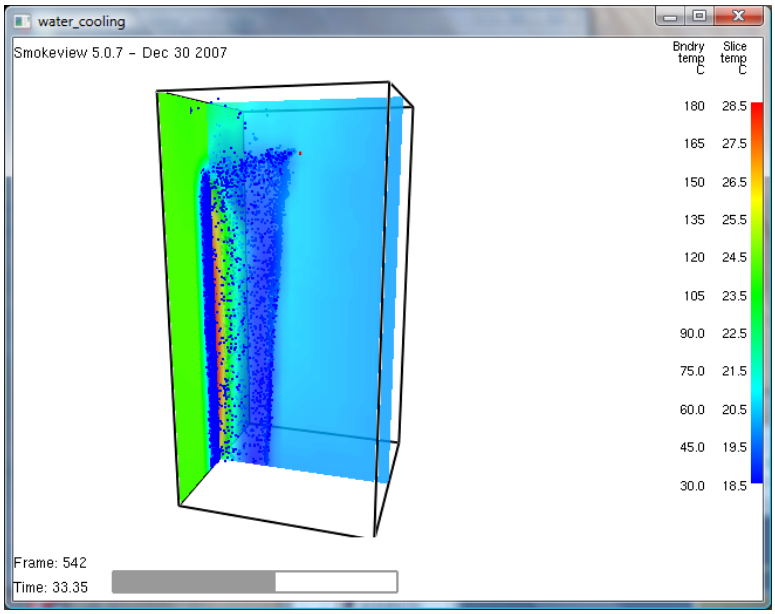

Figure 2.8. Water cooling results

Chapter 3. Burner Fire with Temperature Measurement

In this tutorial you will create a 500 kW burner fire and measure the temperature in the center of the plume at a height of 1.5 m.

This tutorial demonstrates how to:

- Create a burner fire.
- Add a temperature measurement device.
- Add a slice plane for temperature visualization.
- View 3D results using Smokeview.
- View 2D results using PyroSim.

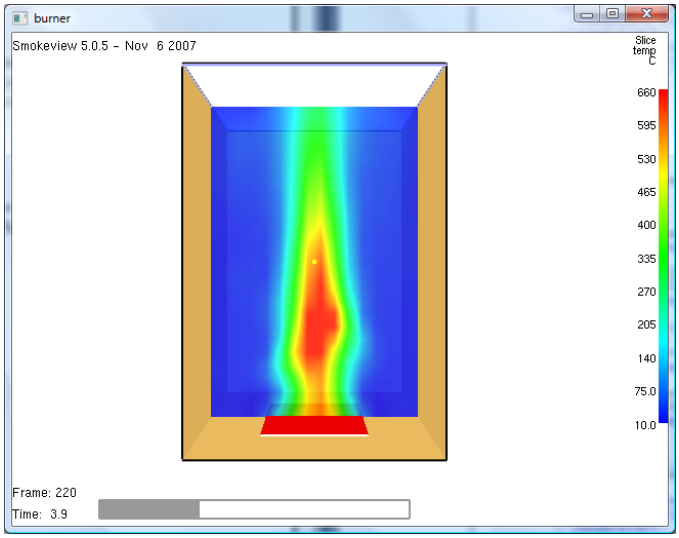

Figure 3.1. Burner fire in this example

Before you begin, ensure that you are using **SI** units (see Chapter 1).

Create the Mesh

In this example we will use mesh cells that are 0.13 m across. This value is approximately 1/5 of the characteristic diameter (D*) for a 500 kW fire. As a rule of thumb, this is as large as the mesh cells can be while still maintaining a moderate level of accuracy in modeling the plume, [U.S. NRC, 2007]. Using mesh cells that are smaller by a factor of 2 should decrease error by a factor of 4, but will increase the simulation run time by a factor of 16.

1. On the **Model** menu, click **Edit Meshes...**.
2. Click **New**, Figure 3.2.
3. In the **Min X** box, type `-1.0` and in the **Max X** box, type `1.0`.

4. In the **Min Y** box, type `-1.0` and in the **Max Y** box, type `1.0`.

5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `3.0`.

6. In the **X Cells** box, type `15`.

7. In the **Y Cells** box, type `15`.

8. In the **Z Cells** box, type `24`.

9. Click **OK** to save changes and close the **Edit Meshes** dialog.

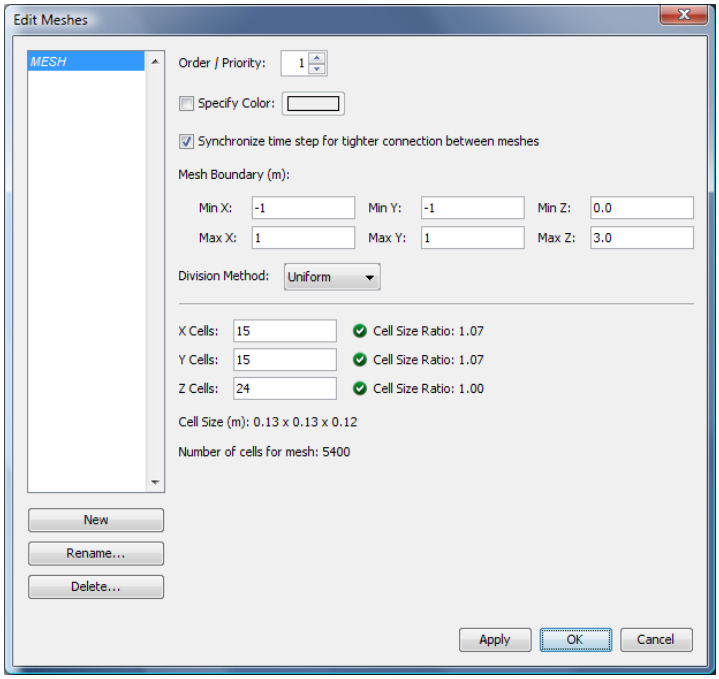

Figure 3.2. Creating the mesh

Create the Burner Surface

Surfaces are used to define the properties of objects in your FDS model. In this example, we define a burner surface that releases heat at a rate of 500 kW/m2.

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New**.

3. In the **Surface Name** box, type `burner`, Figure 3.3.

4. In the **Surface Type** list, select **Burner**.

5. Click **OK** to create the new default burner surface.

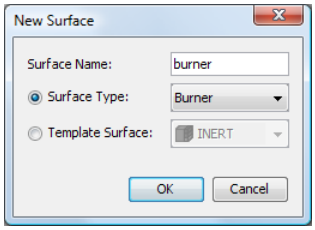

Figure 3.3. Creating a new burner surface

1. In the **Description** box, type `500 kW/m2 burner`, Figure 3.4
2. Click the **Color** button to open the **Surface Color** dialog, then select a good burner color (e.g. red) and click **OK**.
3. In the **Heat Release Rate (HRR)** box, type `500`.
4. Click **OK** to save changes and close the **Edit Surfaces** dialog.

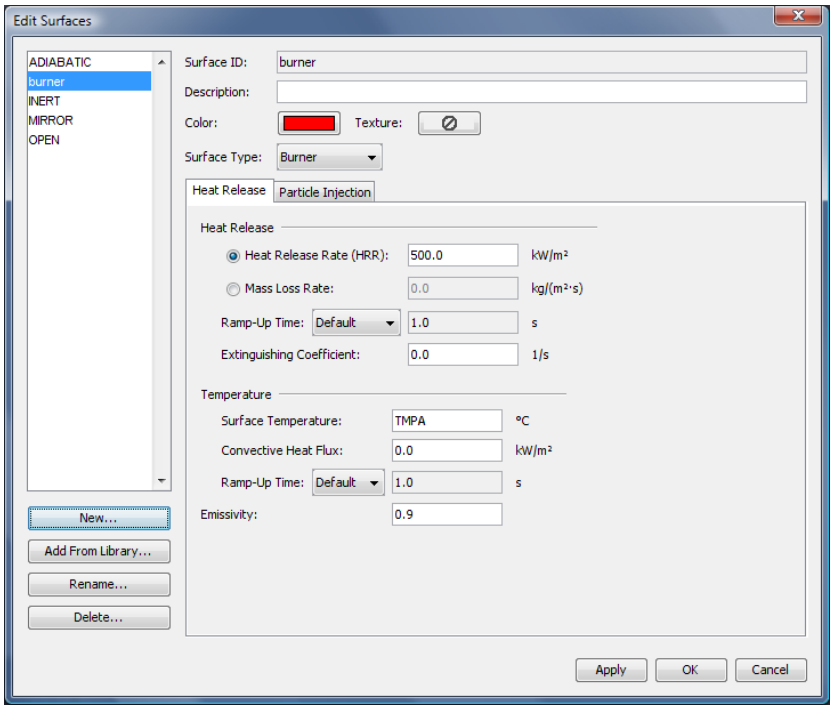

Figure 3.4. Defining parameters for the burner surface

Create the Burner Vent

In this example, we use a vent and the previously created **burner** surface to define the fire.

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `burner vent`, Figure 3.5.
3. In the **Type** list, select **burner**. This specifies that the previously created burner surface will define the properties of the vent.

4. In the **Lies in the Plane** list, select **Z**.

5. In the **Min X** box, type `-0.5` and in the **Max X** box, type `0.5`.

6. In the **Min Y** box, type `-0.5` and in the **Max Y** box, type `0.5`.

7. Click **OK** to create the new burner vent.

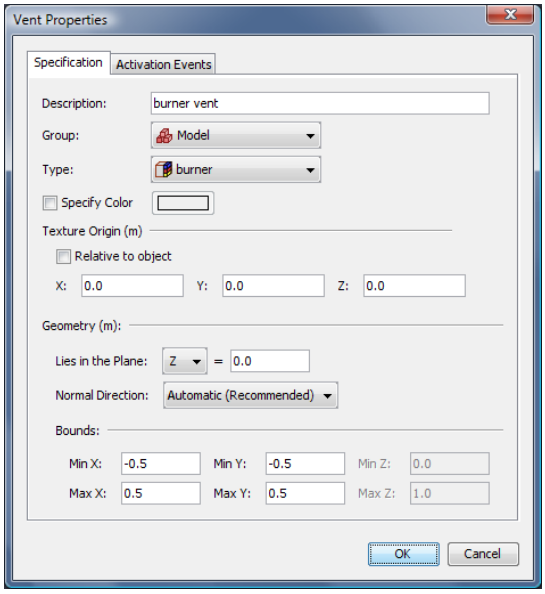

Figure 3.5. Creating the burner vent

Create the Top Vent

The top of the mesh is an open boundary.

1. On the **Model** menu, click **New Vent...**.

2. In the **Description** box, type `open top`.

3. In the **Type** list, select **OPEN**. This is a default surface that means this will be an open boundary.

4. Click the **Specify Color** checkbox and then click the **Color** button to open the **Surface Color** dialog. Select a light blue color and click **OK**.

5. In the **Lies in the Plane** list, select **Z** and type `3.0`.

6. In the **Min X** box, type `-1.0` and in the **Max X** box, type `1.0`.

7. In the **Min Y** box, type `-1.0` and in the **Max Y** box, type `1.0`.

8. Click **OK** to create the open vent.

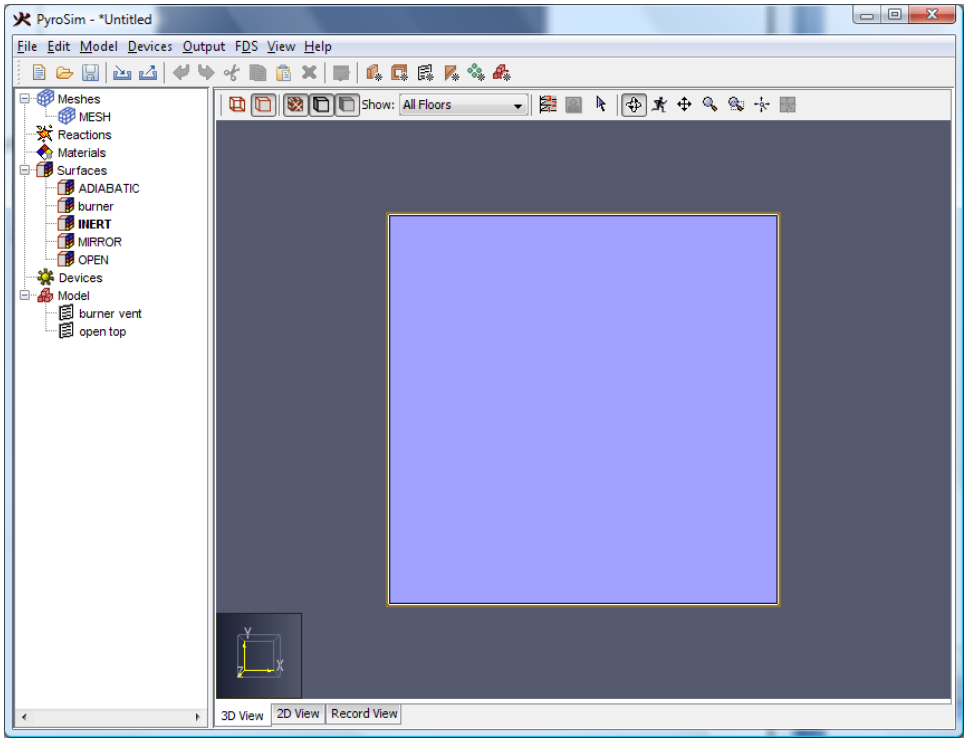

Figure 3.6. After creating the open vent

Rotate the Model for a Better View

1. To reset the zoom and properly center the model, press **CTRL + R**. PyroSim will now be looking straight down at the model along the Z axis.

2. Press the **left mouse button** button in the **3D View** and drag to rotate the model.

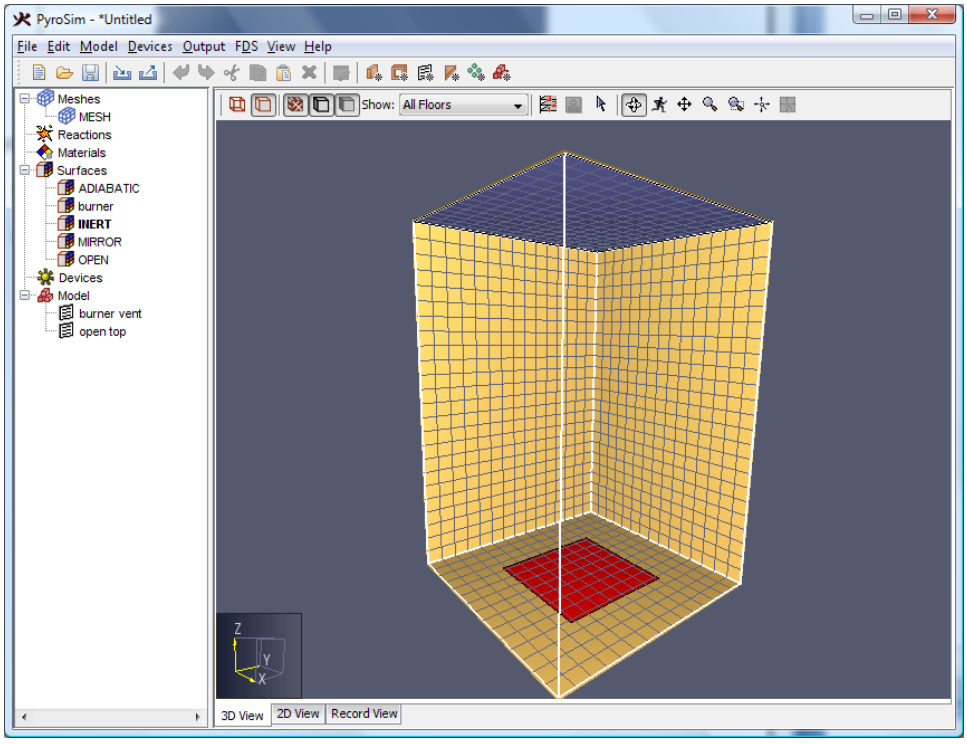

Figure 3.7. The model after rotating. The burner is shown in red and the top vent in blue.

Add a Temperature Measurement Device

1. On the **Devices** menu, click **New Gas-Phase Device...**.
2. In the **Device Name** box, type `temp 1.5 m`.
3. In the **Quantity** list, select **Temperature**.
4. On the **Location** row, in the **Z** box, type `1.5`.
5. Click **OK** to create the temperature measurement device. It will appear as a yellow dot in the center of the model.

Add a Temperature Slice Plane

1. On the **Output** menu, click **Slices...**.
2. In the **XYZ** column, click the cell and select **Y**.
3. In the **Plane Value** column, click the cell and type `0.0`.
4. In the **Gas Phase Quantity** column, click the cell and select **Temperature**.
5. In the **Use Vector?** column, click the cell and select **NO**.
6. Click **OK** to create the slice plane.

Save the Model

1. On the **File** menu, click **Save**.

2. Choose a location to save the model. Because FDS simulations generate many files and a large amount of data, it is a good idea to use a new folder for each simulation. For this example, we will name the file `c:\Burner\burner.psm`..

3. Click **OK** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS...**.

2. Choose a location to save simulation data. For this example, we will name the file `c:\Burner\burner.fds`..

3. Click **OK** to save the FDS input file and begin the simulation.

4. The **FDS Simulation** dialog will appear and display the progress of the simulation. By default, PyroSim specifies a 10 second simulation. This should take approximately 1 minute to run depending on computing hardware, Figure 3.8.

5. When the simulation is complete, Smokeview should launch automatically and display a 3D still image of the model, Figure 3.9..

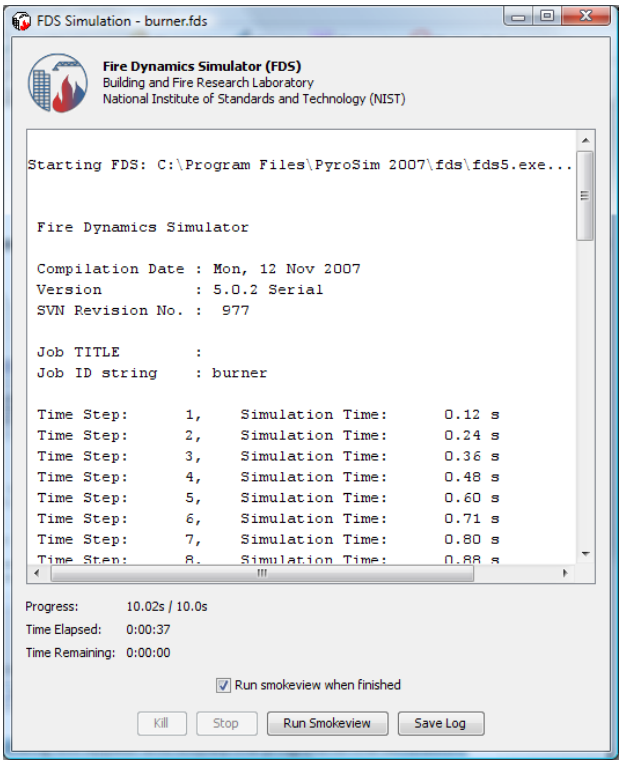
FDS Simulation - burner.fds
Fire Dynamics Simulator (FDS)
Building and Fire Research Laboratory
National Institute of Standards and Technology (NIST)
Starting FDS: C:\Program Files\PyroSim 2007\fds\fds5.exe...
Fire Dynamics Simulator
Compilation Date : Mon, 12 Nov 2007
Version : 5.0.2 Serial
SVN Revision No. : 977
Job TITLE :
Job ID string : burner
Time Step: 1, Simulation Time: 0.12 s
Time Step: 2, Simulation Time: 0.24 s
Time Step: 3, Simulation Time: 0.36 s
Time Step: 4, Simulation Time: 0.48 s
Time Step: 5, Simulation Time: 0.60 s
Time Step: 6, Simulation Time: 0.71 s
Time Step: 7, Simulation Time: 0.80 s
Progress: 10.02s / 10.0s
Time Elapsed: 0:00:37
Time Remaining: 0:00:00
Run smokeview when finished
Kill
Stop
Run Smokeview
Save Log

Figure 3.8. The simulation dialog during the analysis.

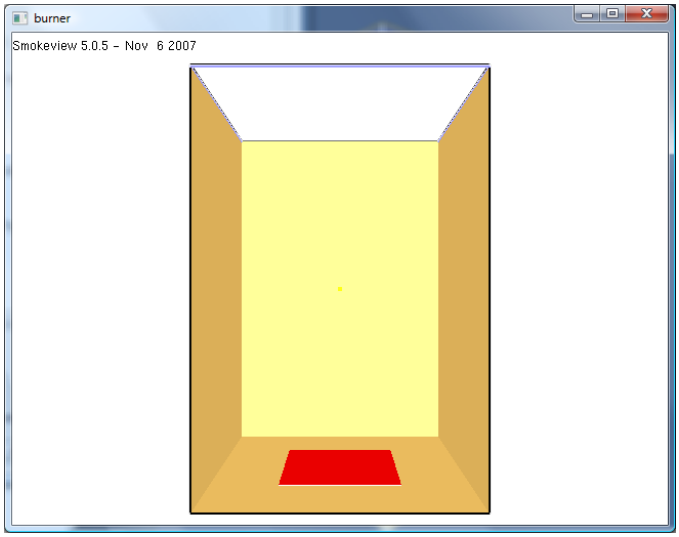

Figure 3.9. The initial Smokeview display.

View Smoke in 3D

1. In the Smokeview window, right-click to activate the menu.
2. In the menu, click **Load/Unload > 3D Smoke > soot mass fraction (RLE)**. This will start an animation of the smoke in this model.
3. To view a specific time in the animation, click the **timeline bar** in the bottom of the Smokeview window. To return to animation mode, press **t**.
4. To reset Smokeview, right-click to activate the menu, then click **Load/Unload > Unload All**.

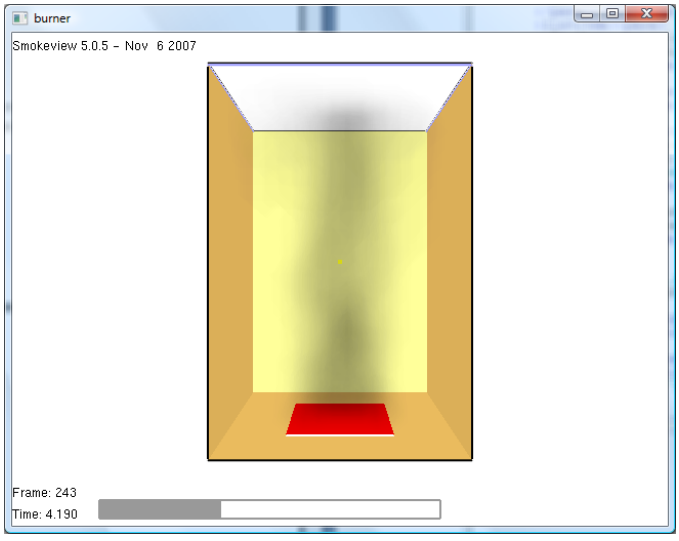

Figure 3.10. 3D smoke in the model.

View Temperature Slice Plane

1. In the Smokeview window, right-click to activate the menu.

2. In the menu, click **Load/Unload > Slice File > TEMPERATURE > Y=0.06667**. This will start an animation of the temperature slice plane. Note that the Y coordinate of the plane was shifted by FDS to correspond to the center of a cell.

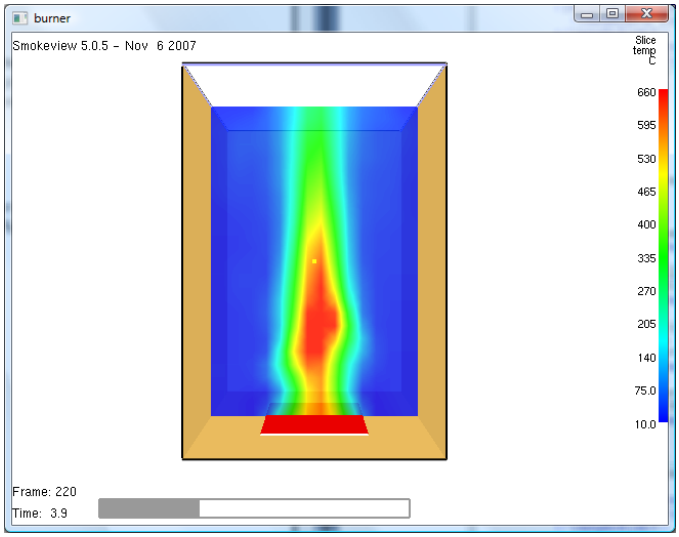

Figure 3.11. Temperature contours on the slice plane.

View Temperature Measurements

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results...**.

2. A dialog will appear showing the different types of 2D results that are available. Select **burner_devc.csv** and click **Open** to view the temperature device output.

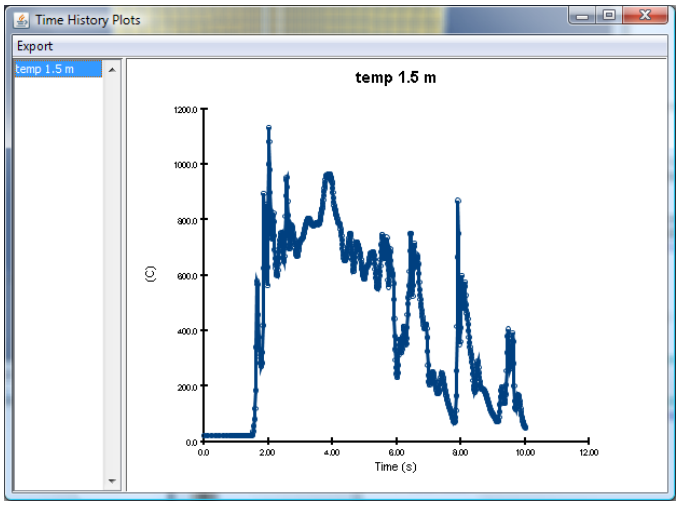

Figure 3.12. Temperature time history plot.

Chapter 4. Smoke Layer Height and Heat Flow Through a Door

In this tutorial you will simulate an 800 kW fire in the corner of a 5m x 5m room. The room has a 1m doorway. You will learn how to measure smoke layer height in the compartment and heat flow though the doorway.

In this tutorial you will:

- Create an 800 kW burner fire.
- Create a doorway using a hole.
- Add a flow measurement device.
- Add a layer zoning device (to measure layer height).
- View 3D results using Smokeview.
- View 2D results using PyroSim.

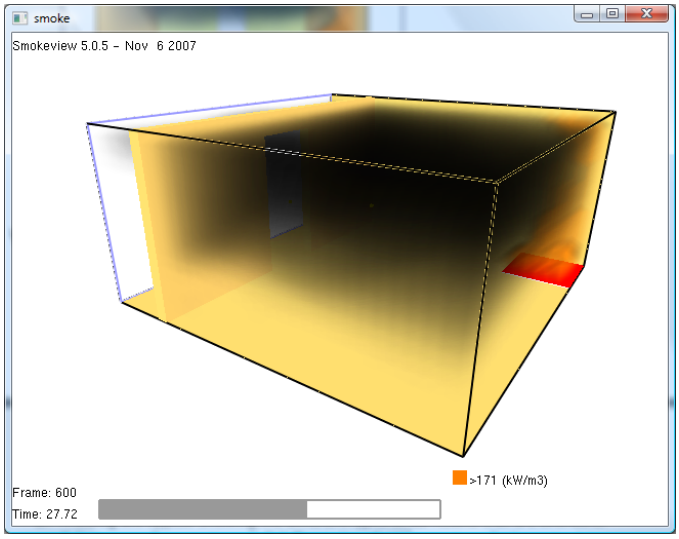

Figure 4.1. Smoke in the model

Before you begin, ensure that you are using **SI** units (see Chapter 1).

Create the Burner Surface

Surfaces are used to define the properties of objects in your FDS model. In this example, we define a burner surface that releases heat at a rate of 800 kW/m2.

1. On the **Model** menu, click **Edit Surfaces...**.
2. Click **New**.
3. In the **Surface Name** box, type `burner`, Figure 4.2.

4. In the **Surface Type** list, select **Burner**.

5. Click **OK** to create the new default burner surface.

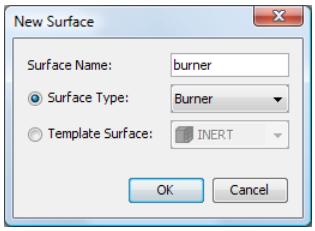

Figure 4.2. Creating a new burner surface

1. In the **Description** box, type `800 kW/m2 burner`, Figure 4.3

2. Click the **Color** button to open the **Surface Color** dialog, then select a good burner color (e.g. red) and click **OK**.

3. In the **Heat Release Rate (HRR)** box, type `800`.

4. Click **OK** to save changes and close the **Edit Surfaces** dialog.

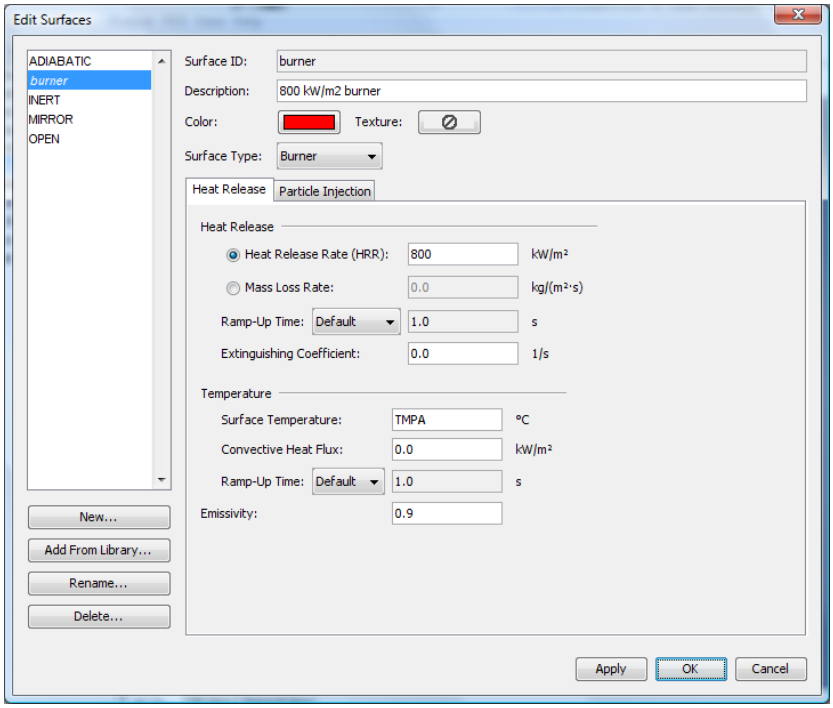

Figure 4.3. Defining parameters for the burner surface

Create the Burner Vent

Vents have general usage in FDS to describe 2D planar objects. Taken literally, a vent can be used to model components of the ventilation system in a building, like a diffuser or a return. In these cases, the vent coordinates define a plane forming the boundary of the duct. No holes need to be created; air is supplied or exhausted by the vent.

You can also use vents as a means of applying a particular boundary condition to a rectangular patch on a surface. A fire, for example, can be created by specifying a vent on either a mesh boundary or solid surface. The vent surface defines the desired characteristics of fire. This is the approach used in this example.

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `burner vent`, Figure 4.4.
3. In the **Type** list, select **burner**. This specifies that the previously created burner surface will define the properties of the vent.
4. In the **Lies in the Plane** list, select **Z**.
5. In the **Min X** box, type `4.0` and in the **Max X** box, type `5.0`.
6. In the **Min Y** box, type `0.0` and in the **Max Y** box, type `1.0`.
7. Click **OK** to create the new burner vent.

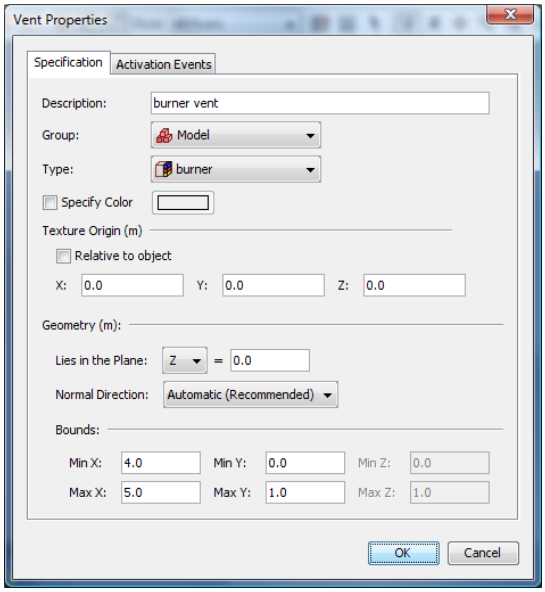

Figure 4.4. Creating the burner vent

Create the Open Side Vent

One side of the model is an open boundary.

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `open side`.
3. In the **Type** list, select **OPEN**. This is a default surface that means this will be an open boundary.
4. Click the **Specify Color** checkbox and then click the **Color** button to open the **Surface Color** dialog. Select a light blue color and click **OK**.

5. In the **Lies in the Plane** list, select **Y** and type `5.0`.

6. In the **Min X** box, type `0.0` and in the **Max X** box, type `5.0`.

7. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.4`.

8. Click **OK** to create the open vent.

Create the Mesh

In this example we will use mesh cells that are 0.17 m across. This value is approximately 1/5 of the characteristic diameter (D*) for a 800 kW fire. As a rule of thumb, this is as large as the mesh cells can be while still maintaining a moderate level of accuracy in modeling the plume, [U.S. NRC, 2007]. Using mesh cells that are smaller by a factor of 2 should decrease error by a factor of 4, but will increase the simulation run time by a factor of 16.

1. On the **Model** menu, click **Edit Meshes...**.

2. Click **New**, Figure 4.5. The boundary dimensions will automatically be set to the correct size based on the two vents.

3. In the **X Cells** box, type `30`.

4. In the **Y Cells** box, type `30`.

5. In the **Z Cells** box, type `15`.

6. Click **OK** to save changes and close the **Edit Meshes** dialog.

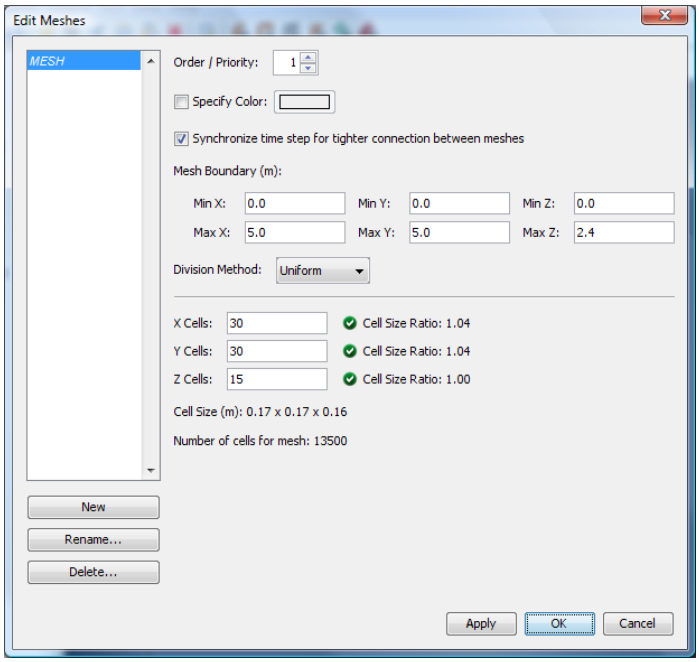

Figure 4.5. Creating the mesh

Add the Wall

In FDS **obstructions** are used to define solid object in the model. In this example, we will use an obstruction to define a wall.

1. On the **Model** menu, click **New Obstruction...**.
2. In the **Description** box, type `wall`, Figure 4.6.
3. In the **Min X** box, type `0.0` and in the **Max X** box, type `5.0`.
4. In the **Min Y** box, type `4.0` and in the **Max Y** box, type `4.2`.
5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.4`.
6. Click **OK** to create the wall obstruction.

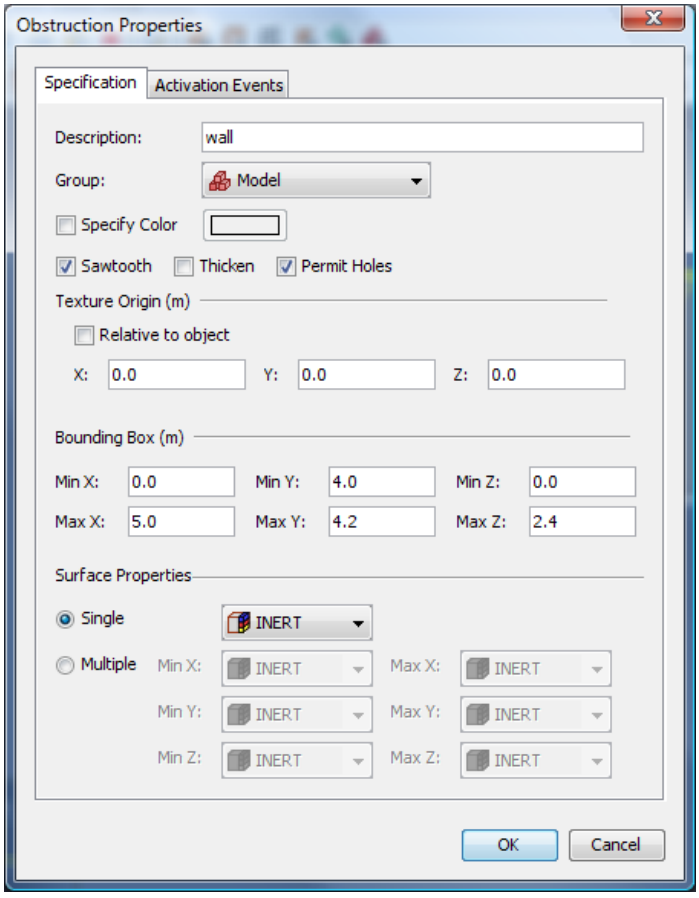

Figure 4.6. Creating the wall

Add the Door

In FDS **holes** are used to define openings through solid objects. In this example, we will use a hole to define a door.

1. On the **Model** menu, click **New Hole...**.
2. In the **Description** box, type `door`.
3. In the **Min X** box, type `2.0` and in the **Max X** box, type `3.0`.
4. In the **Min Y** box, type `3.9` and in the **Max Y** box, type `4.3`.
5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.
6. Click **OK** to create the doorway hole.

Rotate the Model for a Better View

1. To reset the zoom and properly center the model, press **CTRL + R**. PyroSim will now be looking straight down at the model along the Z axis.
2. Press the **left mouse button** button in the **3D View** and drag to rotate the model.

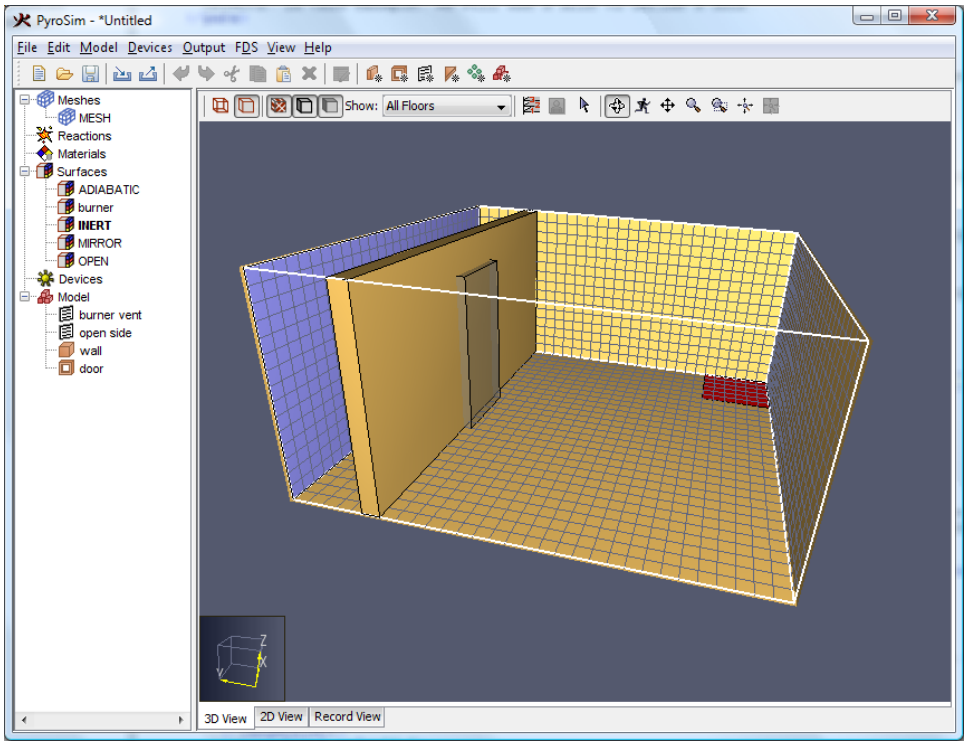

Figure 4.7. The model after rotating. The burner is shown in red and the top vent in blue.

Add a Layer Zoning Device

1. On the **Devices** menu, click **New Layer Zoning Device...**.
2. In the **Device Name** box, type `lz01`.
3. For the **End Point 1** coordinates, in the **X** box, type `2.5`, in the **Y** box, type `2.5`, and in the **Z** box, type `0.0`.

4. For the **End Point 2** coordinates, in the **X** box, type `2.5`, in the **Y** box, type `2.5`, and in the **Z** box, type `2.4`.

5. Click **OK** to create the layer zoning device. It will be displayed as a line in the model.

Add a Flow Measuring Device

1. On the **Devices** menu, click **New Flow Measuring Device...**.

2. In the **Device Name** box, type `door flow`.

3. In the **Plane** list, select **Y** and type `4.0`.

4. In the **Min X** box, type `2.0` and in the **Max X** box, type `3.0`.

5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.

6. Click **OK** to create the flow measuring device. It will appear as a yellow plane in the model.

Set the Simulation Length

1. On the **FDS** menu, click **Simulation Parameters...**.

2. On the **Time** panel, in the **End Time** box, type `45.0`.

3. Click **OK** to save the simulation parameters.

Save the model

1. On the **File** menu, click **Save**.

2. Choose a location to save the model. Because FDS simulations generate many files and a large amount of data, it is a good idea to use a new folder for each simulation. For this example, we will name the file `c:\Smoke\smoke.psm`..

3. Click **OK** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS...**.

2. Choose a location to save simulation data. For this example, we will name the file `c:\Smoke\smoke.fds`..

3. Click **OK** to save the FDS input file and begin the simulation.

4. The **FDS Simulation** dialog will appear and display the progress of the simulation. By default, PyroSim specifies a 10 second simulation. This should take approximately 1 minute to run depending on computing hardware, Figure 4.8.

5. When the simulation is complete, Smokeview should launch automatically and display a 3D still image of the model, Figure 4.9..

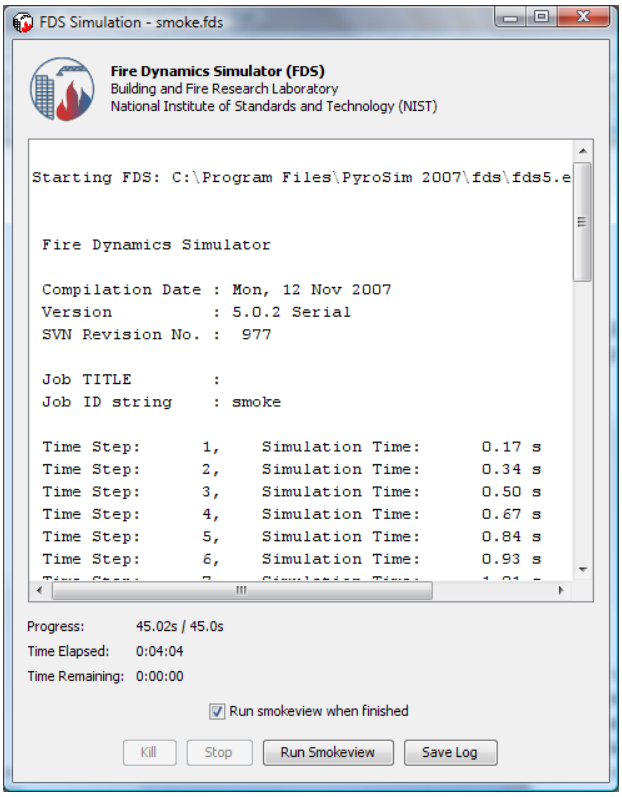
FDS Simulation - smoke.fds
Fire Dynamics Simulator (FDS)
Building and Fire Research Laboratory
National Institute of Standards and Technology (NIST)
Starting FDS: C:\Program Files\PyroSim 2007\fds\fds5.e
Fire Dynamics Simulator
Compilation Date : Mon, 12 Nov 2007
Version : 5.0.2 Serial
SVN Revision No. : 977
Job TITLE :
Job ID string : smoke
Time Step: 1, Simulation Time: 0.17 s
Time Step: 2, Simulation Time: 0.34 s
Time Step: 3, Simulation Time: 0.50 s
Time Step: 4, Simulation Time: 0.67 s
Time Step: 5, Simulation Time: 0.84 s
Time Step: 6, Simulation Time: 0.93 s
Progress: 45.02s / 45.0s
Time Elapsed: 0:04:04
Time Remaining: 0:00:00
Run smokeview when finished
Kill
Stop
Run Smokeview
Save Log

Figure 4.8. The simulation dialog during the analysis.

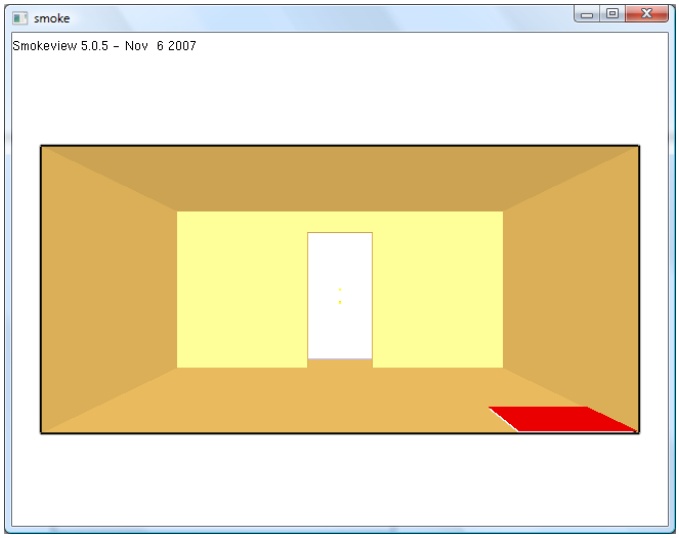

Figure 4.9. The initial Smokeview display.

View Smoke in 3D

1. In the Smokeview window, right-click to activate the menu.

2. In the menu, click **Load/Unload > 3D Smoke > soot mass fraction (RLE)**. This will start an animation of the smoke in this model.

3. In the menu, click **Load/Unload > 3D Smoke > HRRPUV (RLE)**. This will start add an animation of fire to the model in addition to the smoke.

4. To view a specific time in the animation, click the **timeline bar** in the bottom of the Smokeview window. To return to animation mode, press **t**.

5. To reset Smokeview, right-click to activate the menu, then click **Load/Unload > Unload All**.

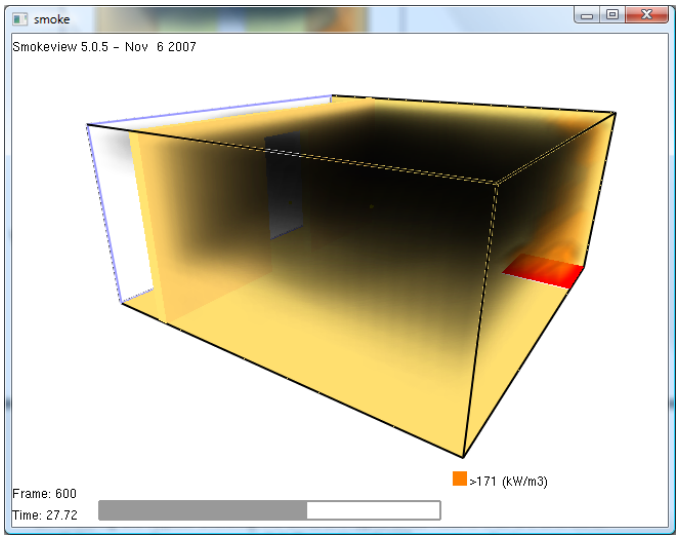

Figure 4.10. 3D smoke in the model.

View Time History Data

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results...**.

2. A dialog will appear showing the different types of 2D results that are available. Select **smoke_devc.csv** and click **Open** to view the device output. The first display will be the heat flow through the door, Figure 4.11.

3. To select a data set from a different sensor, select that data in the left panel. To view smoke layer height data, click **lz01->HEIGHT**, Figure 4.12.

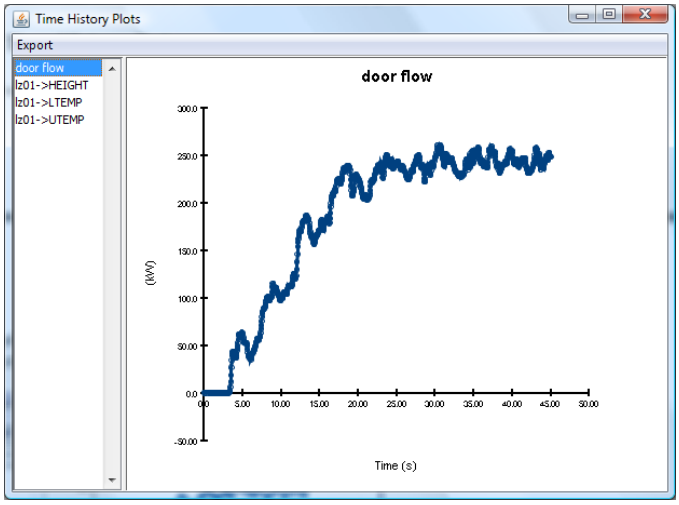

Figure 4.11. Time history plot of heat flow through the door.

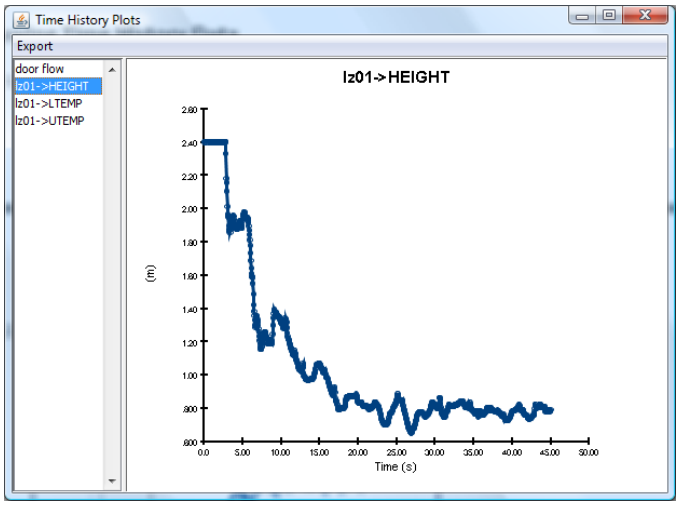

Figure 4.12. Time history plot of heat flow through the door.

Chapter 5. Switchgear Fire Example

This example evaluates fire conditions in two adjacent switchgear rooms connected by a double fire door, Figure 5.1. In the fire scenario, a fire starts in a switchgear cabinet in room 3A. The fire modeling results will used to estimate the time available for operators to conduct manual actions in one of the switchgear rooms. This example was provided by Bryan Klein [Klein, 2007].

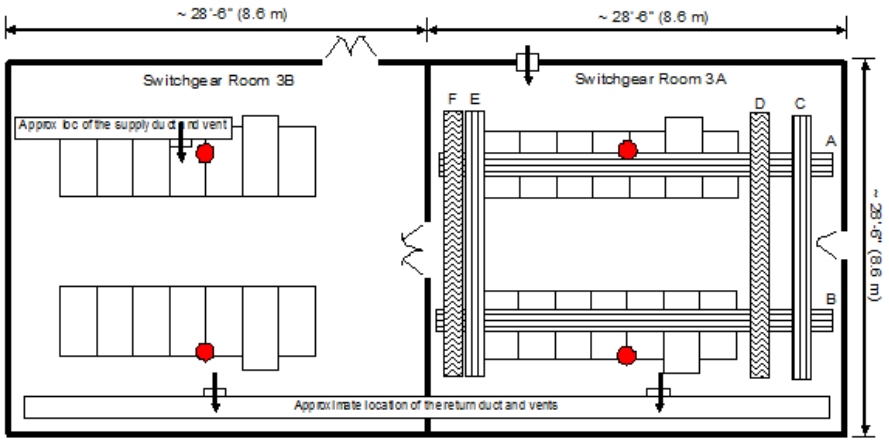

Figure 5.1. Pictorial representation of the switchgear room complex, showing switchgear cabinets, cable trays, supply ducts and vents, and smoke detectors. Drawing not to scale.

This tutorial demonstrates how to:

- Define materials.
- Create and replicate geometry.
- Open doors after a specified time.
- Create a burner fire.
- Add a smoke layer device.
- Add a slice plane for temperature visualization.
- View 3D results using Smokeview.
- View 2D results using PyroSim.

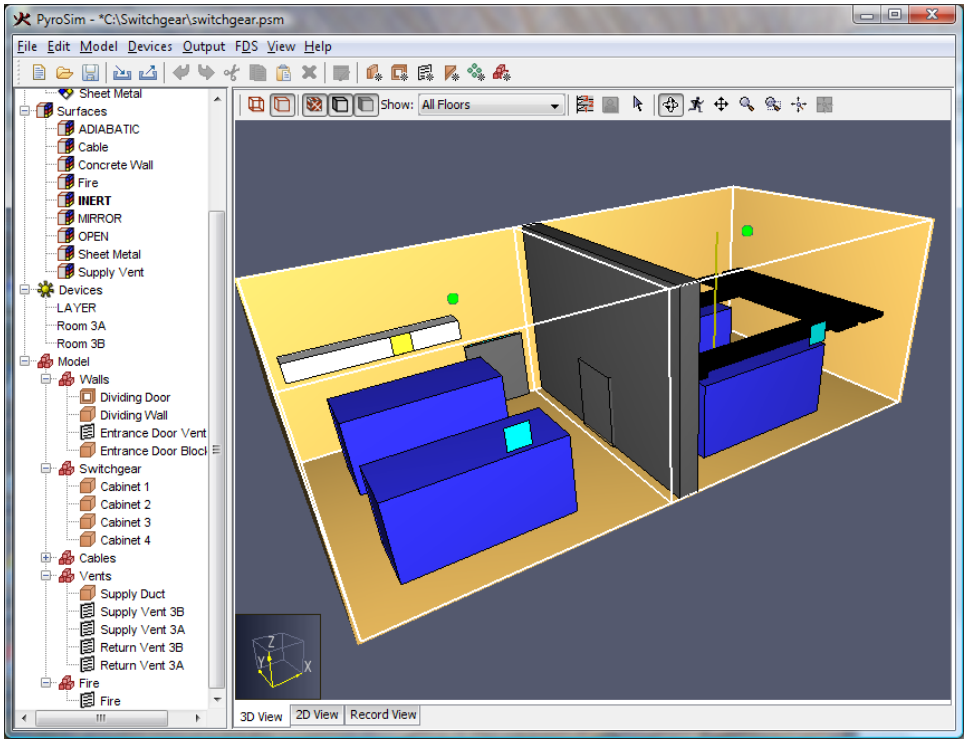

Figure 5.2. Completed model

Model parameters are given below.

Table 5.1. Room size (interior dimensions)

Dimension	English	Metric
Length	28'-6"	8.6 m
Width	28'-6"	8.6 m
Height	20'	6.0 m
Wall Thickness	2'	0.6096 m

Table 5.2. Door size

Dimension	English	Metric
Width	3'	0.9 m
Height	8'	2.4 m

Table 5.3. Concrete properties (NUREG/CR-1805)

Property	Value
Density	2400 kg/m^3
Specific Heat	0.75 kJ/kg-K
Conductivity	1.6 W/m-K

Table 5.4. Sheet metal properties (generic)

Property	Value
Density	7850 kg/m^3
Specific Heat	0.45 kJ/kg-K
Conductivity	46.6 W/m-K

Table 5.5. Cable properties (NUREG/CR-6850)

Property	Value
Density	1380 kg/m^3
Specific Heat	1.289 kJ/kg-K
Conductivity	0.192 W/m-K

Computational Mesh

In this example, we will use two meshes. We will use relatively coarse meshes that should be refined for a final analysis. In Room 3A (the room on the right) the cell size will be approximately 0.5 ft (0.1524 m) and in Room 3B, approximately 1.0 ft (0..3048 m). We have selected a finer resolution in Room 3A to more accurately represent the geometry of the cable trays and to provide a finer resolution for the flow solution near the fire. The two meshes much touch in order to transfer information between them. We will position the common plane inside Room 3B, so that the finer mesh includes all of Room 3A and the door between the rooms.

There is always a compromise between number of cells and acceptable solution time. As described, this model will have 162 000 cells and run in approximately 8 hours on a single CPU computer.

This problem uses English units as the primary values for the geometry. Switch to English units

1. On the **View** menu, click **Units**.
2. Select **English**.

To create the first solution mesh for Room 3A

1. On the **Model** menu, click **Edit Meshes...**.
2. Click **New** to create a mesh.
3. Click **Rename** and in the **Name** box, type `Room 3A`. Click **OK** to close the rename dialog.
4. In the **Order/Priority** list, select `1`. This ensures that the finer mesh is the primary mesh for the solution.
5. In the mesh boundary boxes, enter the values in Table 5.6.
6. In the **X, Y, and Z** cell boxes, enter `60`, enter `60`, and enter `40` respectively, as shown in Figure 5.3. The FDS solution is optimized when the mesh cell division is defined by a number

that can be formed using multiples of powers of 2, 3 and 5. These divisions give a cell size of approximately 0.5 ft (0.1524 m).

7. Click **Apply** to create the mesh.

Table 5.6. Dimensions for the mesh in Room 3A (including 2' thick walls)

	X (ft)	Y (ft)	Z (ft)
Min	27.5	0.0	0.0
Max	59.0	28.5	20.0

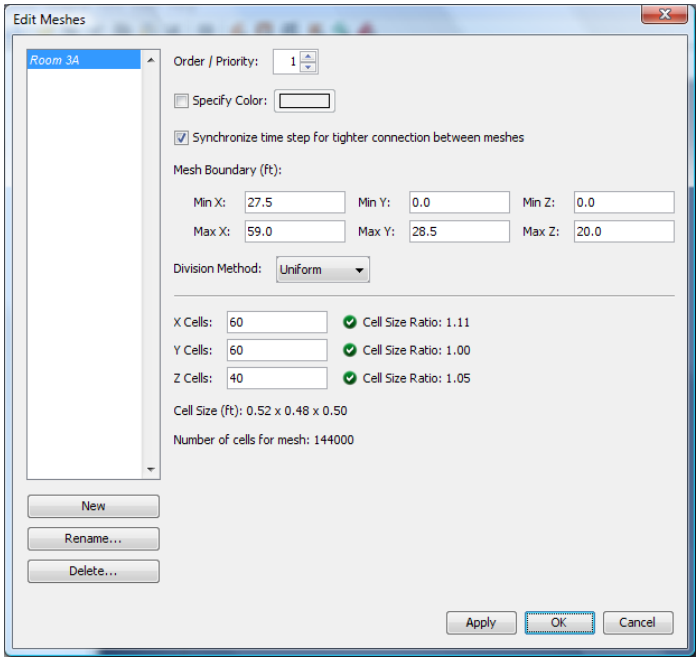

Figure 5.3. Input to create the mesh

To create the second solution mesh for Room 3B

1. Click **New** to create a mesh.

2. Click **Rename** and in the **Name** box, type `GRID Room 3B`. Click **OK** to close the rename dialog.

3. In the **Order/Priority** list, select `2`.

4. In the mesh boundary boxes, enter the values in Table 5.7.

5. In the **X, Y, and Z** cell boxes, enter `30`, enter `30`, and enter `20` respectively. These divisions give a cell size of approximately 1.0 ft (0.3048 m).

6. Click **OK** to save the data and create the meshes.

Table 5.7. Dimensions for the mesh in Room 3B (including 2' thick walls)

	X (ft)	Y (ft)	Z (ft)
Min	0.0	0.0	0.0
Max	27.5	28.5	20.0

The meshes are shown in Figure 5.4. On the toolbar, click to reset the image. Click . You can rotate, pan, and zoom the model using the mouse and the **Shift** and **Alt** keys.

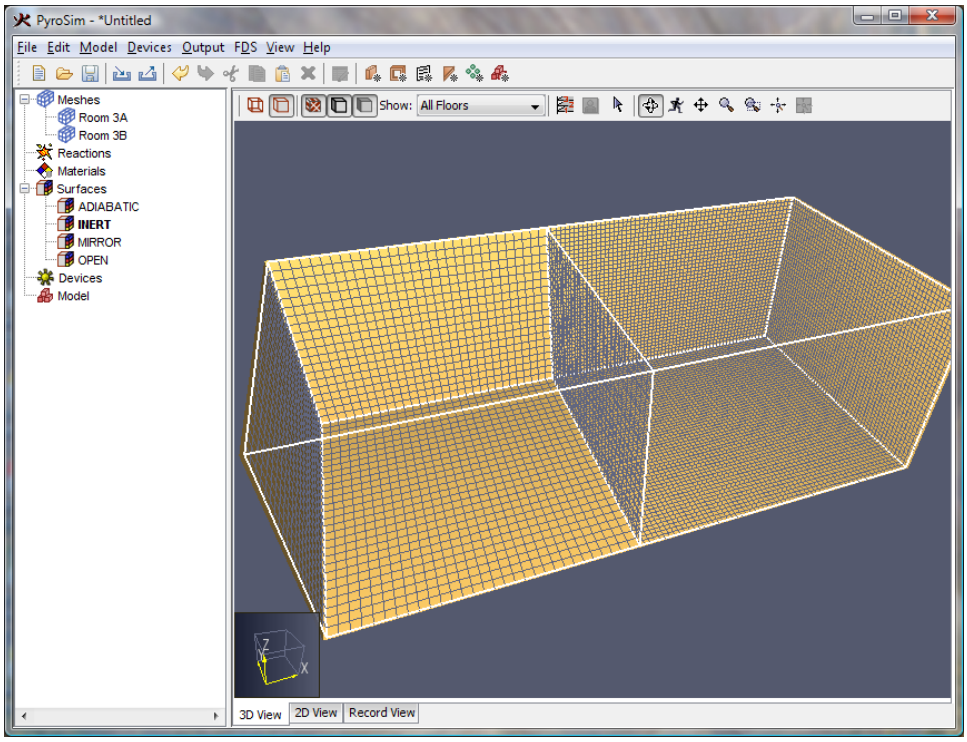

Figure 5.4. Display of the mesh

Material Properties

FDS Version 5 uses materials to define physical properties. In this model, we will include the following material types: concrete, sheet metal, and thermo-plastic cable. We note that the material properties in the problem description have been provided in metric units, so we will temporarily switch to metric units.

1. On the **View** menu, click **Units**.

2. Select **SI**.

To add the concrete to our model

1. On the **Model** menu, click **Edit Materials...**.

2. Click **New...**.

3. In the **Material Name** box, type `Concrete`, in the **Material Type** list, select `Solid`. Click **OK** to close the **New Material** dialog.

4. In the **Density** box, type `2400` kg/m^3,

5. In the **Specific Heat** box, type `0.75` kJ/kg-K,

6. In the **Conductivity** box, type `1.6` W/m-K,

7. Click **Apply** to save the changes, Figure 5.5.

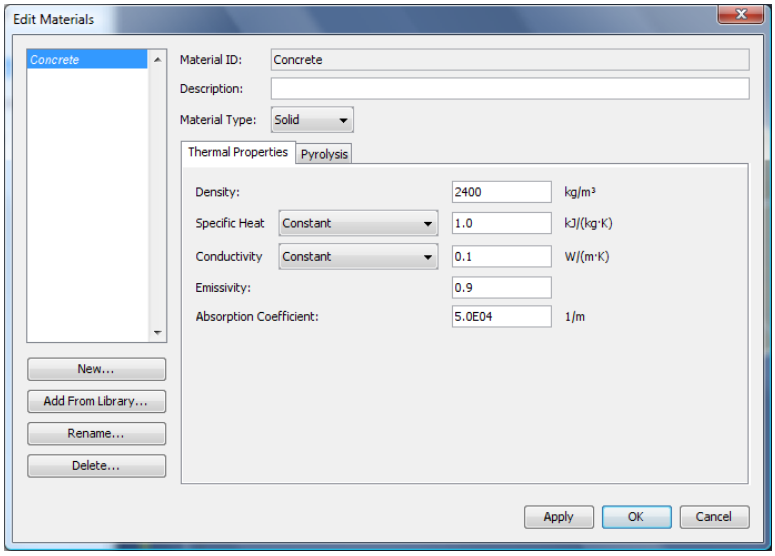

Figure 5.5. Concrete properties

To add steel

1. In the **Edit Materials...**dialog, click **New...**.

2. In the **Material Name** box, type `Steel`, in the **Material Type** list, select `Solid`. Click **OK** to close the **New Material** dialog.

3. In the **Density** box, type `7850` kg/m^3,

4. In the **Specific Heat** box, type `0.45` kJ/kg-K,

5. In the **Conductivity** box, type `46.6` W/m-K,

6. Click **Apply** to save the changes.

The cables will be represented as a thermo-plastic material

1. In the **Edit Materials...**dialog, click **New...**.

2. In the **Material Name** box, type `Cable`, in the **Material Type** list, select `Solid`. Click **OK** to close the **New Material** dialog.

3. In the **Density** box, type `1380` kg/m^3,

4. In the **Specific Heat** box, type `1.289` kJ/kg-K,

5. In the **Conductivity** box, type `0.192` W/m-K,

6. Click **OK** to save the changes and close the **Edit Materials** dialog.

Save the Model

This is a good time to save the model.

1. On the **File** menu, click **Save**.

2. Choose a location to save the model. Because FDS simulations generate many files and a large amount of data, it is a good idea to use a new folder for each simulation. For this example, we will name the file `C:\Switchgear\switchgear.psm`..

3. Click **OK** to save the model.

Surface Properties

Surfaces are used to define the properties of objects in your model. Solid surfaces will use the material properties previously defined. Vent and burner will be defined directly, without reference to materials.

This problem uses English units as the primary values for the geometry. Switch to English units

1. On the **View** menu, click **Units**.

2. Select **English**.

The concrete wall is 2 ft (0.6096 m) thick. This will be a **Layered** surface, although we will define only one layer. To create this surface

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New...**.

3. In the **Surface Name** box, type `Concrete Wall`, in the **Surface Type** list, select `Layered`. Click **OK** to close the **New Surface** dialog.

4. Click the **Color** button to open the **Surface Color** dialog, then select a dark gray and click **OK**.

5. In the **Material Layers** panel, in the **Thickness** column, type `2.0` ft.

6. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction**column, type `1.0`. In the **Material**column, select `Concrete`, Figure 5.6. Click **OK** to close the **Composition** dialog.

7. In the **Edit Surfaces** dialog, click **Apply** to save the changes, Figure 5.7.

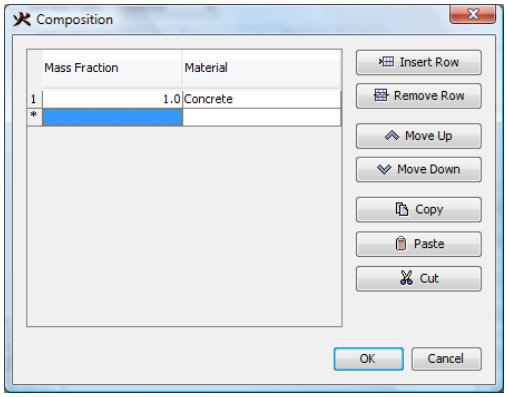

Figure 5.6. Composition of the concrete surface

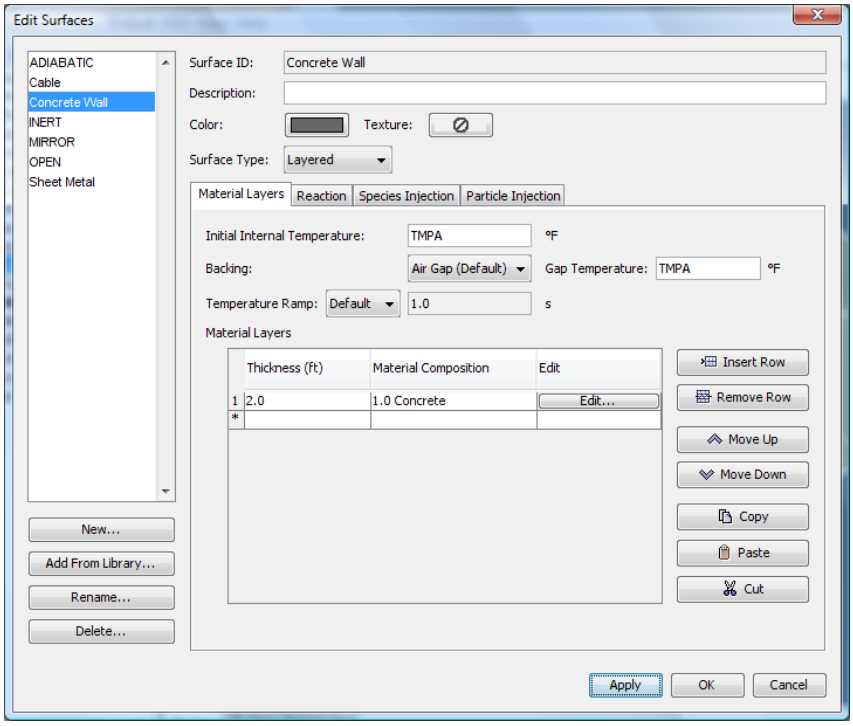

Figure 5.7. The concrete surface

The sheet metal is 0.05 in (0.0042 ft, 0.0013 m) thick. To create this surface

1. In the **Edit Surfaces** dialog, click **New...**.
2. In the **Surface Name** box, type `Sheet Metal`, in the **Surface Type** list, select `Layered`. Click **OK** to close the **New Surface** dialog.
3. Click the **Color** button to open the **Surface Color** dialog, then select blue and click **OK**.
4. In the **Material Layers** panel, in the **Thickness** column, type `0.0042` ft.
5. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction**column, type `1.0`. In the **Material**column, select `Steel`. Click **OK** to close the **Composition** dialog.

6. In the **Edit Surfaces** dialog, click **Apply** to save the changes.

The cable is 1.5 in (0.125 ft, 0.0381 m) thick. To create this surface

1. In the **Edit Surfaces** dialog, click **New...**.
2. In the **Surface Name** box, type `Cable`, in the **Surface Type** list, select `Layered`. Click **OK** to close the **New Surface** dialog.
3. Click the **Color** button to open the **Surface Color** dialog, then select a black and click **OK**.
4. In the **Material Layers** tab, in the **Backing** list, select `Insulated`. This will approximate a solid being heated from all sides.
5. In the **Material Layers** panel, in the **Thickness** column, type `0.125` ft.
6. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction**column, type `1.0`. In the **Material**column, select `Cable`. Click **OK** to close the **Composition** dialog.
7. Click on the **Reaction** tab.
8. Click to select **Governed Manually**.
9. In the **Heat Release Rate** box, type `265` kW/m^2. In the **Ignition** panel select **Ignite at** and type `250` C.
10. Click **OK** to save the changes and close the **Edit Surfaces** dialog.

Model Geometry

We will now create the geometry using a few dialogs and by drawing a few walls. If you were working on a more complex building, you could display the floor plan in the 2D View and then sketch walls directly on the background image or import the data from a DXF file.

The model will only include the separating wall between the two rooms. There is no need to define the perimeter walls, since the perimeter will be defined to have the surface properties of the concrete wall.

During the creation of a PyroSim model, it is recommended that you group the various parts of the building for purposes of editing and later viewing. For example, all walls will be placed in a sub-group called **Walls**. This allows us to quickly select this group to modify it or to hide it for viewing of other interior details.

The problem dimensions use English units as the primary values. To switch to English units

1. On the **View** menu, click **Units**.
2. Select **English**.

Walls and Doors

To make a new group

1. On the **Model** menu, click **New Group...**.
2. In the **Parent Group** list, select `Model`.
3. In the **Group Name** box, type `Walls`.
4. Click **OK** to close the **Create Group** dialog.

To define the dividing wall

1. On the **Model** menu, click **New Obstruction...**.
2. In the **Description** box, type `Dividing Wall`.
3. In the **Group** list, select `Walls`.
4. In the **Bounding Box** boxes, enter the values in Table 5.8, Figure 5.8.
5. In the **Surface Properties** panel, select **Single** and select **Concrete Wall** from the list.
6. Click **OK** to close the **Obstruction Properties** dialog.
7. On the **View** menu, click un-select the **Show Grids** option. The display should now look like Figure 5.9.

Table 5.8. Dividing wall dimensions

	X (ft)	Y (ft)	Z (ft)
Min	28.5	0.0	0.0
Max	30.5	28.5	20.0

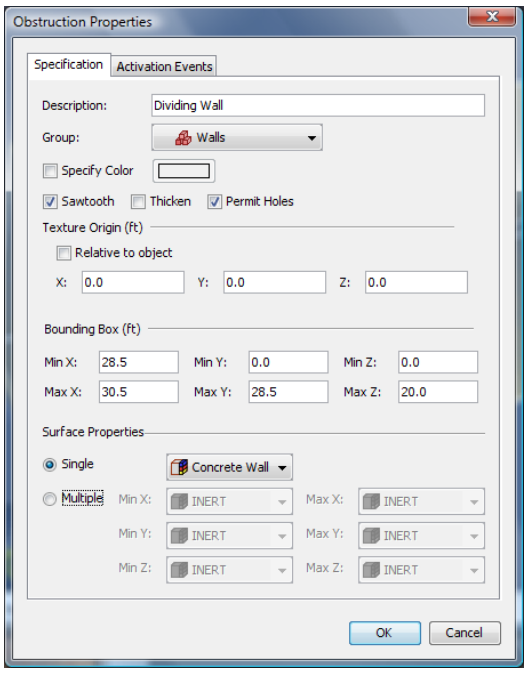

Figure 5.8. Input for the wall dividing the two rooms

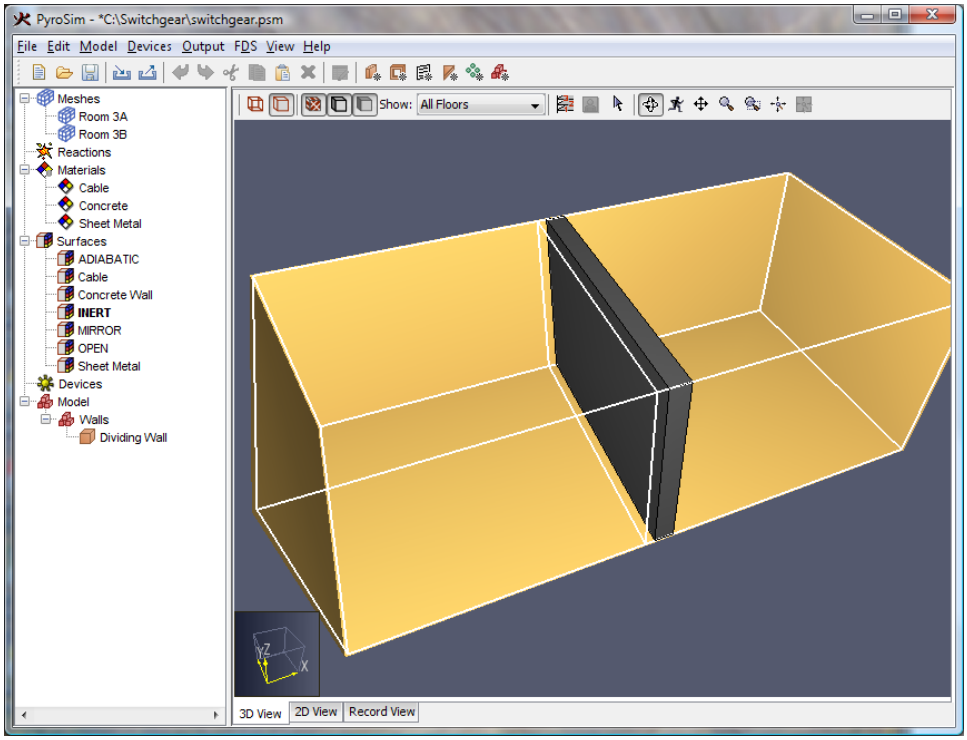

Figure 5.9. Display of the dividing wall

We will use a **Hole** obstruction to represent the door. A hole can defined in the model, but not activate until a specified time or event. In this case, we will define a hole (door) between the rooms that will open 5 minutes (300 seconds) after the start of the fire. To add the dividing door to the model

1. On the **Model** menu, click **New Hole...**.
2. In the **Description** box, type `Dividing Door`.
3. In the **Group** list, select `Walls`.
4. In the **Bounding Box** boxes, enter the values in Table 5.9. The values are chosen thicker than the wall to ensure that the door takes precedence.
5. Click the **Activation Events** tab.
6. In the **Time** box, type `300`. In the **Event** list, select `Activate`. The hole will be created (door open) 300 seconds after the fire starts.
7. Click **OK** to close the **Aligned Hole Properties** dialog.

Table 5.9. Dividing door dimensions

	X (ft)	Y (ft)	Z (ft)
Min	28.25	11.25	0.0
Max	30.75	17.25	8.0

We also will add the Room 3B entrance door. We want this door to stay closed for 300 sec. By default, FDS assumes the boundary of a mesh is closed. To define a door on the boundary, we will first create an open vent on the mesh boundary, then plug it with an obstruction that will be removed at 300 sec. To add the entrance door to the model

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `Entrance Door Vent`.
3. In the **Group** list, select `Walls`.
4. In the **Type** list, select `OPEN`.
5. In the **Lies in the Plane** list, select **Y** and type `28.5`.
6. In the **Min X** box, type `20.0` and in the **Max X** box, type `26.0`.
7. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `8.0`.
8. Click **OK** to save the changes and close the **Vent Properties** dialog.

To define the blockage in the entrance door

1. On the **Model** menu, click **New Obstruction...**.
2. In the **Description** box, type `Entrance Door Blockage`.
3. In the **Group** list, select `Walls`.
4. In the **Bounding Box** boxes, enter the values in Table 5.10. The values are chosen thicker than the wall to ensure that the door takes precedence.
5. In the **Surface Properties** panel, select **Single** and select **Concrete Wall** from the list.
6. Click the **Activation Events** tab.
7. In the **Time** box, type `300`. In the **Event** list, select `Deactivate`. The blockage will be created (door open) 300 seconds after the fire starts.
8. Click **OK** to close the **Obstruction Properties** dialog.

Table 5.10. Entrance door blockage dimensions

	X (ft)	Y (ft)	Z (ft)
Min	19.5	28.0	0.0
Max	26.5	29.0	8.0

Switchgear Cabinets

First, make a switchgear group

1. On the **Model** menu, click **New Group...**.

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Switchgear`.

4. Click **OK** to close the **Create Group** dialog.

We could define the cabinets using the **New Obstruction...** menu and dialog as described above. Instead we will sketch the cabinets. To prepare to draw the switchgear cabinets

1. Click **2D View**.

2. On the **View** menu, click **Select Grid** and select the `Sketch Grid`. The **Sketch Grid** allows you to specify a grid spacing that is useful for creating geometry independent of the solution meshes.

3. On the **View** menu, click **Set Sketch Grid Spacing...**. In the **Snap Spacing** box, type `0.5`. Click **OK** to close the **Snap-to Grid** dialog.

4. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

5. To set the tool properties, click **Tool Properties** ( ).

6. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `8.0`.

7. In the **Surface Prop** list, select **Sheet Metal**.

8. Click **OK** to close the **New Obstruction Properties** dialog.

9. In the **2D View** action toolbar (top of window), select **Switchgear** in the **Group** list.

To draw the cabinet

1. Press and drag the mouse, approximating the dimensions of the lower left cabinet shown in Figure 5.1.

2. Your sketch will approximate that shown in Figure 5.10.

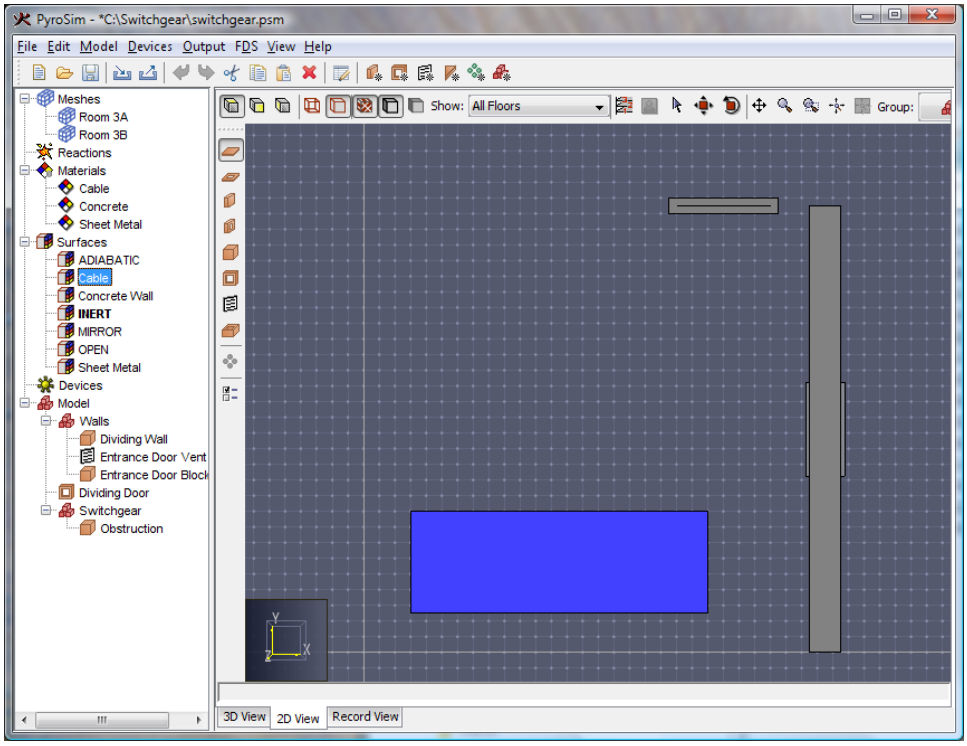

Figure 5.10. Sketch of the lower left cabinet

By default, the cabinet sketch snapped to the sketch grid points. To precisely define the dimensions of the cabinet

1. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool ( ).

2. Double-click on the cabinet to edit its properties.

3. In the **Description** box, type `Cabinet 1`.

4. In the **Bounding Box** boxes, enter the values in Table 5.11.

5. Click **OK** to close the **Obstruction Properties** dialog.

Table 5.11. Cabinet Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	4.0	4.0	0.0
Max	20.0	10.0	8.0

We will now copy the first cabinet using the **Translate Objects** tool in the **2D View**.

1. Using the **Select and Manipulate Objects** tool ( ), click on **Cabinet 1**.

2. Click the **Translate Objects** tool ( ).

3. Press the **CTRL** key (to create a copy) and drag the vertical handle to create a second cabinet located symmetrically relative to **Cabinet 1**, Figure 5.11.

4. Click the **Select and Manipulate Objects** tool ( ).

5. Double click on the new cabinet and change the name to **Cabinet 2**. The dimensions for Cabinet 2 are given in Table 5.12.

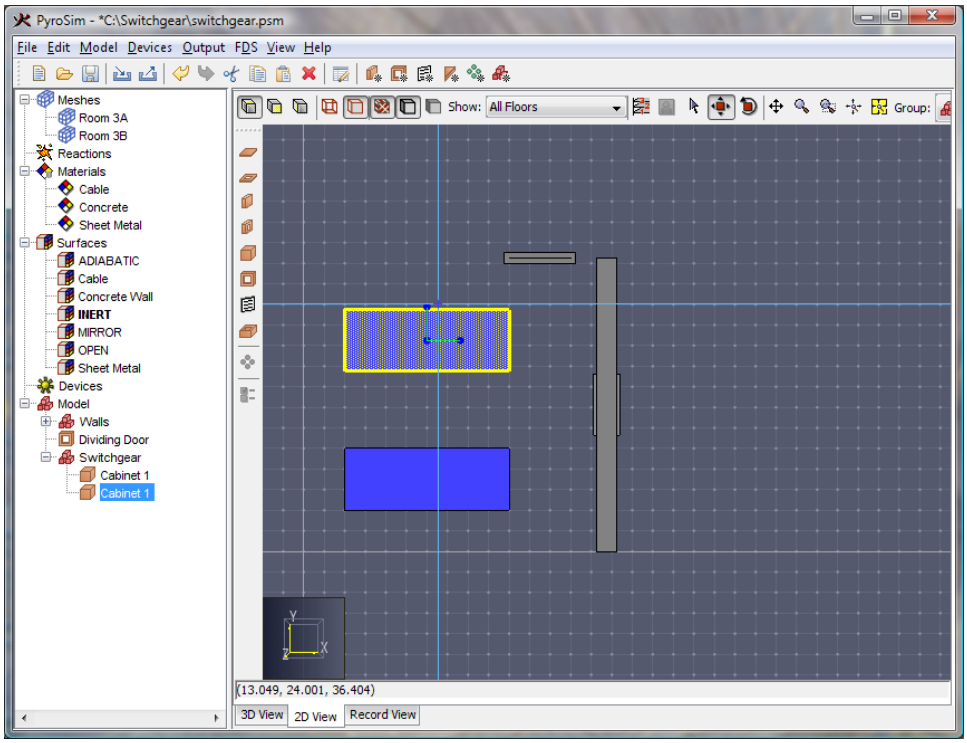

Figure 5.11. Making a copy of Cabinet 1 by dragging the vertical handle. The final position will be 4 feet from the left and top boundaries.

Table 5.12. Dimensions for Cabinet 2

	X (ft)	Y (ft)	Z (ft)
Min	4.0	18.5	0.0
Max	20.0	24.5	8.0

We now copy both cabinets using the **Copy/Move...** menu. You could alternately use the **Translate Objects** tool as before.

1. Using the **Select and Manipulate Objects** tool ( ), press **CTRL** and click on both **Cabinet 1** and **Cabinet 2**.

2. Right-click on either cabinet and click **Copy/Move...**.

3. Click **Copy** and type `1` in the **Number of Copies** box.

4. In the **Offset** box for **X**, type `30.5` .

5. Click **OK**.

6. Change the cabinet names to **Cabinet 3** and **Cabinet 4**.

The cabinets are shown in Figure 5.12. Right-click on the **Bottom Wall** and click **Hide Object(s)** to see inside the room. You can always select **Show All Objects** to turn on an object display.

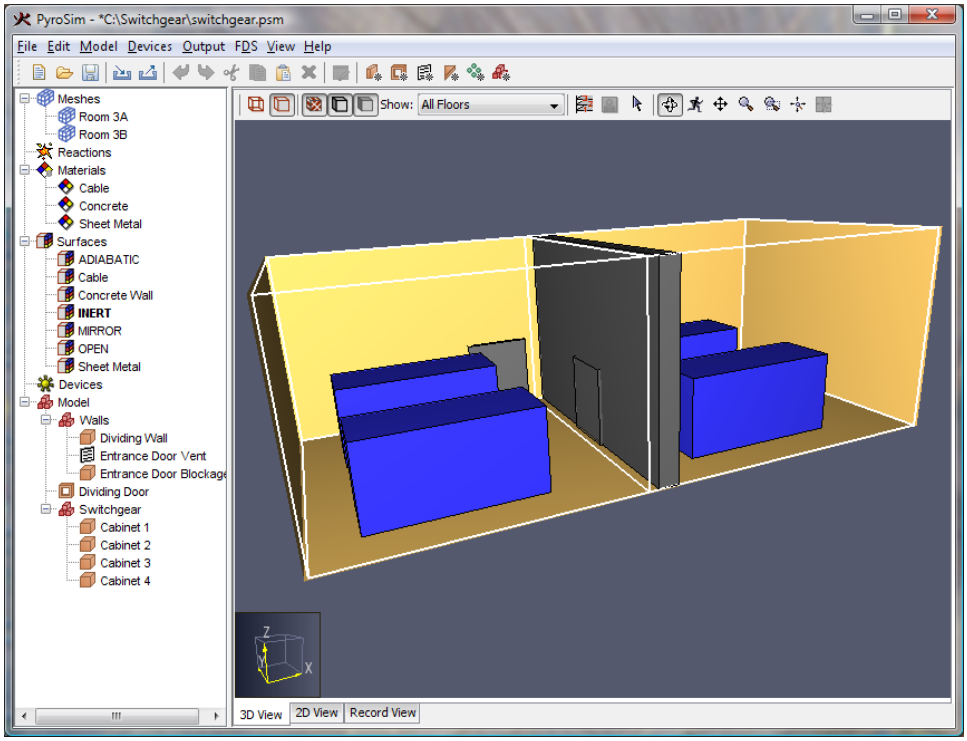

Figure 5.12. The rooms showing the switchgear cabinets

Cable Trays

The cables are supported by ladder trays. These trays are relatively open to air flow and have relatively little thermal mass compared to the cables. Therefore, we will not model the details of the trays, but will just model the cables. First, make a cables group

1. On the **Model** menu, click **New Group...**.

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Cables`.

4. Click **OK** to close the **Create Group** dialog.

To prepare to draw the cables

1. Click **2D View**. This will display the sketch grid.

2. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

3. To set the tool properties, click **Tool Properties** ( ).
4. In the **Min Z** box, type `8.5`.
5. In the **Max Z** box, type `9.0`.
6. In the **Surface Prop** list, select **Cable**.
7. Click **OK** to close the **New Obstruction Properties** dialog.
8. In the **2D View** action toolbar (top of window), select **Cables** in the **Group** list.

To draw cable A

1. Press and drag the mouse, approximating the dimensions of cable A shown in Figure 5.1.
2. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool.
3. Double-click on the cable to edit its properties.
4. In the **Description** box, type `Cable A`.
5. In the **Bounding Box** boxes, enter the values in Table 5.13.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 5.13. Cable A Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	31.0	20.0	8.5
Max	58.0	23.0	9.0

Copy to create cable B

1. Right-click on **Cable A** and click **Copy/Move...**.
2. Click **Copy** and type `1` in the **Number of Copies** box.
3. In the **Offset** box for **Y**, type `-14.5`.
4. Click **OK** to create a **Cable A[1]**.
5. Double click and change the name to **Cable B**.

Drawing cables C, D is similar. First

1. Click **2D View**.
2. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

3. To set the tool properties, click **Tool Properties** ( ).
4. In the **Min Z** box, type `9.5`.
5. In the **Max Z** box, type `10.0`.
6. In the **Surface Prop** list, select **Cable**.
7. Click **OK** to close the **New Obstruction Properties** dialog.
8. In the **2D View** action toolbar (top of window), select **Cables** in the **Group** list.

To draw cable C

1. Press and drag the mouse, approximating the dimensions of cable C shown in Figure 5.1.
2. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool.
3. Double-click on the cable to edit its properties.
4. In the **Description** box, type `Cable C`.
5. In the **Bounding Box** boxes, enter the values in Table 5.14.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 5.14. Cable C Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	55.0	3.0	9.5
Max	58.0	25.5	10.0

Copy to create cable D

1. Right-click on **Cable C** and click **Copy/Move...**.
2. Click **Copy** and type `1` in the **Number of Copies** box.
3. In the **Offset** box for **X**, type `-4.0` .
4. Click **OK**. This will create **Cable C[1]**. Double click and change the name to **Cable D**.

Copy to create cable E

1. Right-click on **Cable D** and click **Copy/Move...**.
2. Click **Copy** and type `1` in the **Number of Copies** box.
3. In the **Offset** box for **X**, type `-20.0` .
4. Click **OK**. This will create **Cable D[1]**. Double click and change the name to **Cable E**.

If wanted, you could add additional cables. We will stop here. The cables are shown in Figure 5.13.

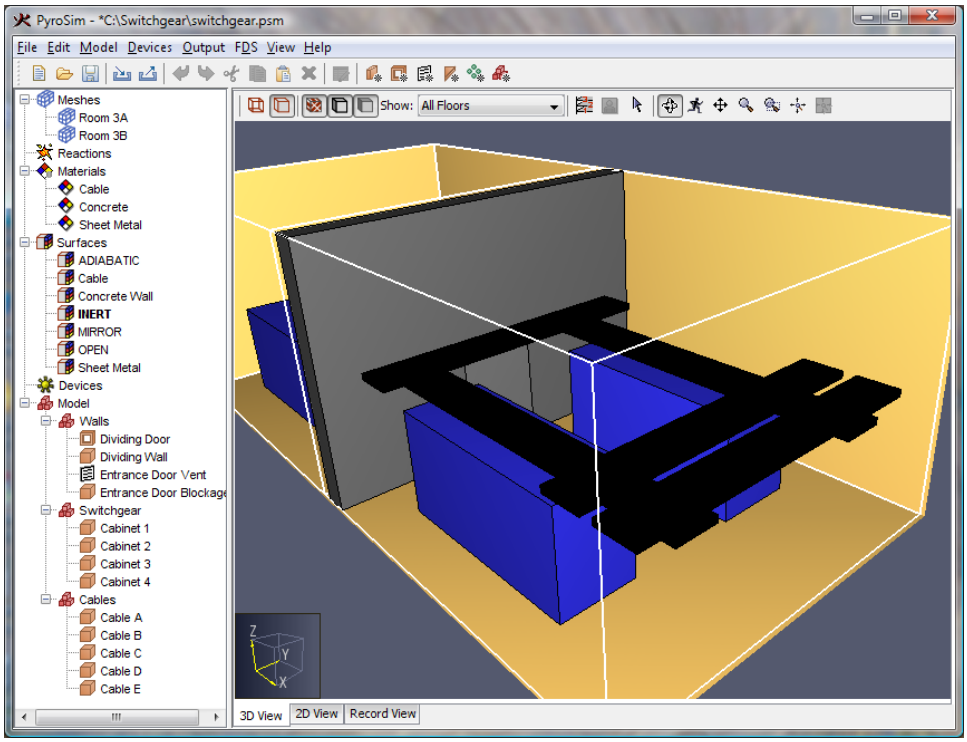

Figure 5.13. The room showing the cables

Ventilation

Each room has supply and return ventilation. Vents have general usage in FDS to describe 2D planar objects. In this case, we will use vents to model components of the ventilation system. No holes need to be created; it is assumed that air is supplied or returned into duct work within the wall.

We will first create a vent group

1. On the **Model** menu, click **New Group...**.

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Vents`.

4. Click **OK** to close the **Create Group** dialog.

There are three steps to defining a vent. First, a surface will be created that defines the flow boundary conditions. Next an obstruction is created. Finally, the vent (using the vent surface), is attached to the obstruction. We will assume all vents are 24"x24" (2'x2'). For the supply vents, we will define a flow rate. For the return vents, we will just have the vents open to the atmosphere. This ensures that the air pressure in the room is not affected by any difference between supply and return flow rates.

To create the surface that represents air supply

1. On the **Model** menu, click **Edit Surfaces...**.
2. In the **Edit Surfaces** dialog, click **New...**.
3. In the **Surface Name** box, type `Supply Vent`.
4. Select **Surface Type** and in the list, select `Supply`.
5. Click **OK** to close the **New Surface** dialog.

To change the properties of the **Supply Vent**

1. In the **Edit Surfaces** dialog, click to select **Supply Vent**.
2. Click the **Color** button to open the **Surface Color** dialog, then select a yellow and click **OK**.
3. On the **Air Flow** tab, click **Specify Velocity** and type `8.33` ft/s, Figure 5.14. This corresponds to 2000 CFM flow through a 2'x2' duct.
4. Click the **Particle Injection** tab.
5. Click the **Emit Particles** checkbox.
6. Click **OK** to save the changes and close the **Edit Surfaces** dialog.

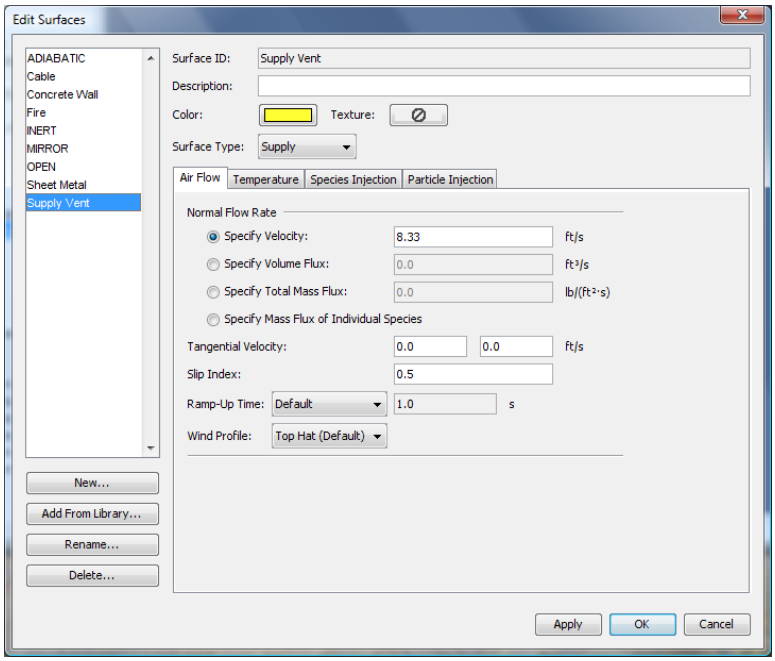

Figure 5.14. Creating the supply vent surface

In this model, we will not explicitly represent the return duct. Instead, we will position the return vents on the outer boundary. This has the advantage of simplicity and allows us to define these vents as open to the atmosphere. To add the return vent to the model

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `Return Vent 3B`.
3. In the **Group** list, select `Vents`.
4. In the **Type** list, select `OPEN`.
5. In the **Lies in the plane** list, select `Y` and type `0.0`. This places it just barely outside the duct for visualization. A word of caution. In most cases this slight offset will not cause any problem, since both the vent and duct will be moved to the same mesh coordinates during the solution. However, it can happen that the small offset will mean that the vent is moved to a different mesh coordinate than the duct. An unattached vent will be ignored in the solution, so either: position the vent directly on the duct or look at the mesh in Smokeview to make sure you see the vent displayed.
6. In the **Bounds** boxes, enter the values in Table 5.15.
7. Click **OK** to close the **Vent Properties** dialog.

Table 5.15. Return vent dimensions

	X (ft)	Z (ft)
Min	13.0	11.0
Max	15.0	13.0

Copy to create the second return vent

1. In the **Navigation View** (tree on left), right-click on **Return Vent 3B** and click **Copy/Move...**.
2. Click **Copy** and type `1` in the **Number of Copies** box.
3. In the **Offset** box for **X**, type `30.5` .
4. Click **OK**. This will create **Return Vent 3B[1]**.
5. Double click and change the name to **Return Vent 3A**.

To create the supply duct for Room 3B

1. On the **Model** menu, click **New Obstruction...**.
2. In the **Description** box, type `Supply Duct`.
3. In the **Group** list, select `Vents`.
4. Click **Specify Color**. The default gray color does not need to be changed.
5. In the **Bounding Box** boxes, enter the values in Table 5.16.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 5.16. Supply duct dimensions

	X (ft)	Y (ft)	Z (ft)
Min	1.0	23.5	11.0
Max	18.0	25.5	13.0

Add the supply vent to the supply duct in Room 3B

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `Supply Vent 3B`.
3. In the **Group** list, select `Vents`.
4. In the **Type** list, select `Supply Vent`.
5. In the **Lies in the plane** list, select `Y` and type `23.49`. This places it just barely outside the duct for visualization, but it will be returned to the exact mesh coordinates during solution.
6. In the **Bounds** boxes, enter the values in Table 5.17.
7. Click **OK** to close the **Vent Properties** dialog.

Table 5.17. Supply vent dimensions

	X (ft)	Z (ft)
Min	11.0	11.0
Max	13.0	13.0

Add the supply vent to the wall in Room 3A

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `Supply Vent 3A`.
3. In the **Group** list, select `Vents`.
4. In the **Type** list, select `Supply Vent`.
5. In the **Lies in the plane** list, select `Y` and type `28.5`.
6. In the **Bounds** boxes, enter the values in Table 5.18.
7. Click **OK** to close the **Vent Properties** dialog.

Table 5.18. Return vent dimensions

	X (ft)	Z (ft)
Min	34.0	11.0
Max	36.0	13.0

An image showing the ducts and vents is given in Figure 5.15. The duct color has been changed to cyan.

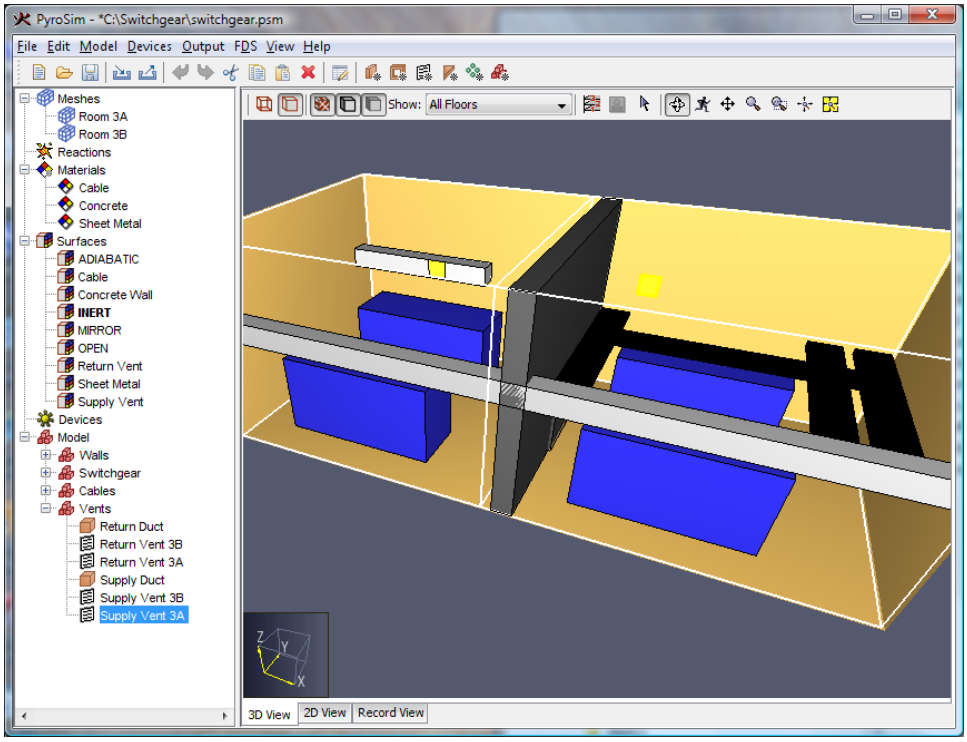

Figure 5.15. The room showing the vents

Fire

The fire is postulated to start in Room 3A in a switchgear cabinet. Heat is released to the room through a vent in the side of the cabinet. The size of the vent is 3.0'x1.0' and located 0.5 ft below the top of the cabinet. We will model the fire as a surface with a specified heat release rate. A surface type is used to specify the fire.

We will first create a fire group

1. On the **Model** menu, click **New Group...**.

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Fire`.

4. Click **OK** to close the **Create Group** dialog.

There are two additional steps to defining a fire. First, a surface will be created that defines the heat release rate. Next, the fire will be attached to the top of the switchgear cabinet. Since the fire properties are specified in SI units

1. On the **View** menu, click **Units**.

2. Select **SI**.

To create the surface that represents fire

1. On the **Model** menu, click **Edit Surfaces...**.
2. In the **Edit Surfaces** dialog, click **New...**.
3. In the **Surface Name** box, type `Fire`.
4. Select **Surface Type** and in the list, select `Burner`.
5. Click **OK** to close the **New Surface** dialog.

To change the properties of the **Fire**

1. In the **Edit Surfaces** dialog, click to select **Fire**.
2. Click the **Color** button to open the **Surface Color** dialog, then select red and click **OK**.
3. Select **Heat Release Rate** box and type `3000.0` kW/m^2. Note that this value is the HRR per unit area. This corresponds to a peak HRR of 834 kW total distributed over an area of 3.0 square feet (0.278 square meters).
4. In the **Ramp-up Time** list, select `T2` and type a value of `60` s. This means the fire will ramp up over 60 seconds to the peak value.
5. Click **OK** to close the **Edit Surfaces** dialog.

Switch back to English units to define the geometry.

1. On the **View** menu, click **Units**.
2. Select **English**.

To add the fire to the top of the cabinet

1. On the **Model** menu, click **New Vent...**.
2. In the **Description** box, type `Fire`.
3. In the **Group** list, select `Fire`.
4. In the **Type** list, select `Fire`.
5. In the **Lies in the plane** list, select `X` and type `50.51`. This places it just barely outside the cabinet for visualization, but it will be returned to the exact mesh coordinates during solution. To repeat a word of caution. In most cases this slight offset will not cause any problem, since both the vent (fire) and cabinet will be moved to the same mesh coordinates during the solution. However, it can happen that the small offset will mean that the vent is moved to a different mesh coordinate than the cabinet. An unattached vent will be ignored in the solution, so either: position the vent directly on the cabinet or look at the mesh in Smokeview to make sure you see the vent (fire) displayed.
6. In the **Bounds** boxes, enter the values in Table 5.19.

7. Click **OK** to close the **Vent Properties** dialog.

Table 5.19. Fire dimensions

	Y (ft)	Z (ft)
Min	20.0	6.5
Max	23.0	7.5

Smoke Detectors

Each room has a smoke detectors. In this analysis, they will be used to output data, but will not activate any features in the model.

To create the smoke detectors

1. On the **Devices** menu, click **New Smoke Detector...**.
2. In the **Detector Name** box, type `Room 3B`.
3. In the **Location** boxes, type `14.0`, `10.0`, and `19.75`.
4. Click **OK** to create the detector.
5. On the **Devices** menu, click **New Smoke Detector...**.
6. In the **Detector Name** box, type `Room 3A`.
7. In the **Location** boxes, type `44.5`, `10.0`, and `19.75`.
8. Click **OK** to create the detector.

Post-Processing Controls

There are a number of tools provided in PyroSim to instrument the building and determine what is going on at each time step. Slice files, thermocouples, and boundary files are some of the most common instruments. We will define a layer device.T.

Layer Zone Device

To add a layer zone device

1. On the **Devices** menu, click **New Layer Zoning Device...**.
2. In the **Device Name** box, type `Layer Room 3A`.
3. To add the device to Room 3A, fill the **Path** table by entering the values in Table 5.20.
4. Click **OK** to close the **Layer Zoning Device** dialog.

Repeat these steps, but this time adding a device to Room 3B, using the data in Table 5.21.

Table 5.20. Layer data for Room 3A

	X (ft)	Y (ft)	Z (ft)
End Point 1 (ft)	44.25	14.25	1.0
End Point 2 (ft)	44.25	14.25	19.0

Table 5.21. Layer data for Room 3B

	X (ft)	Y (ft)	Z (ft)
End Point 1 (ft)	14.25	14.25	1.0
End Point 2 (ft)	14.25	14.25	19.0

Slice Planes

Slice planes can be used to display 2D contours in the Smokeview display of the results. In this analysis, we will save temperature data for future plotting. We will define a slice plane through the center of the room and the fire. To define the slice planes

1. On the **Output** menu, click **Slices...**.
2. Fill the table by entering the values in Table 5.22. You can click on the row number to select entire rows to copy and paste, speeding the entry.
3. Click **OK** to close the **Animated Planar Slices** dialog.

Table 5.22. Slice plane data

XYZ Plane	Plane Value (ft)	Gas Phase Quantity	Use Vector?
Y	14.25 ft	Temperature	NO
Y	21.5	Temperature ft	NO
Z	19.0 ft	Temperature	NO

Boundary (Surface) Plots

Temperatures (and other quantities) on the surface of objects can be plotted by selecting *Surface Plots*. To define the surface plots

1. On the **Output** menu, click **Boundary Quantities...**.
2. Click the **Wall Temperature** checkbox.
3. Click **OK** to close the **Animated Boundary Quantities** dialog.

Isosurfaces

Isosurfaces are 3D plots that show the surface on which a quantity has a specified value. We will save isosurface data for temperatures, which is one way to define the location of a hot gas layer. To define the isosurfaces

1. On the **Output** menu, click **Isosurfaces...**.
2. Click the **Temperature** checkbox.
3. In the **Contour Values** column, enter `50 100 150 200 250`, where the contour values are separated by a space, Figure 5.16
4. Click **OK** to close the **Animated Isosurfaces** dialog.

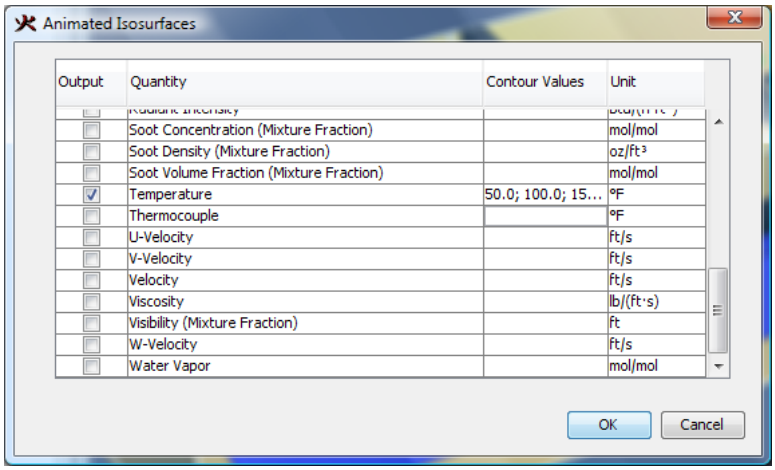

Figure 5.16. Defining the temperature isosurfaces

Simulation Parameters

To define the end time

1. On the **FDS** menu, click **Simulation Parameters...**.
2. In the **Simulation Title** box, type `Switchgear Fire`.
3. In the **End Time** box, type `600` s.

To define the surface to be used on the mesh boundary

1. On the **Simulation Parameters** dialog, click the **Misc.** tab.
2. In the **Default Surface Type** list, select `Concrete Wall`.
3. Click **OK** to close the **Simulation Parameters** dialog.

Run the Analysis

Your model should now look like Figure 5.17. Save the model. To run the analysis

1. On the **FDS** menu, click **Run FDS...**.

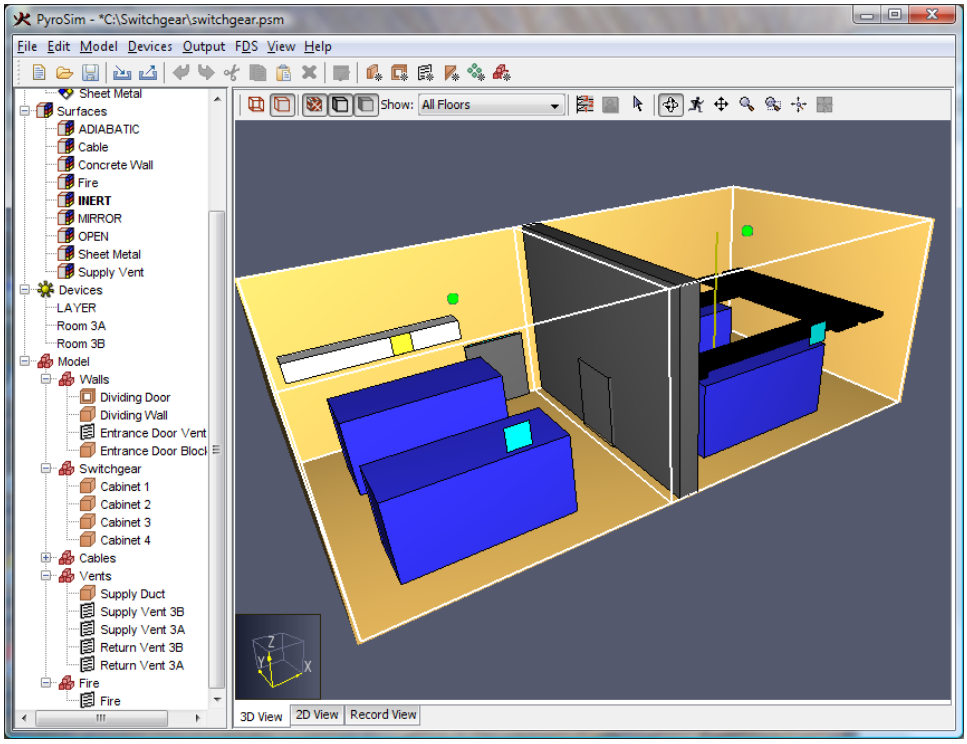

Figure 5.17. Completed model

View the Results

You can use the SmokeView software to make plots of the results. An image showing the smoke density is shown in Figure 5.18. An image showing the temperature contours on the slice planes is shown in Figure 5.19. An image showing the temperature contours on the surfaces is shown in Figure 5.20.

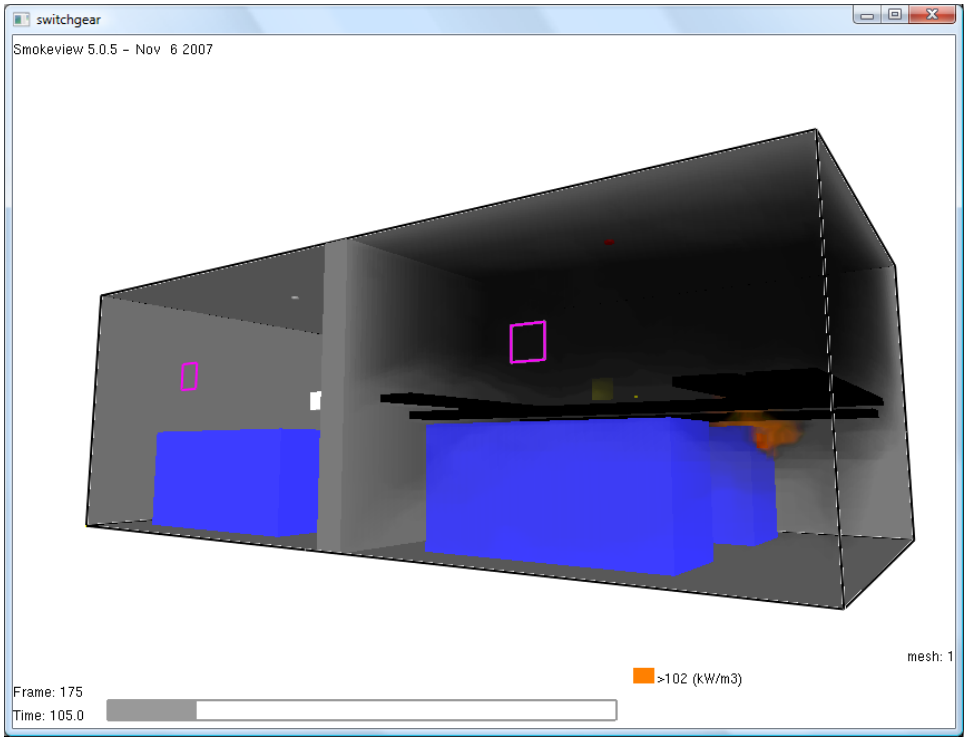

Figure 5.18. Smoke in rooms at 105 seconds

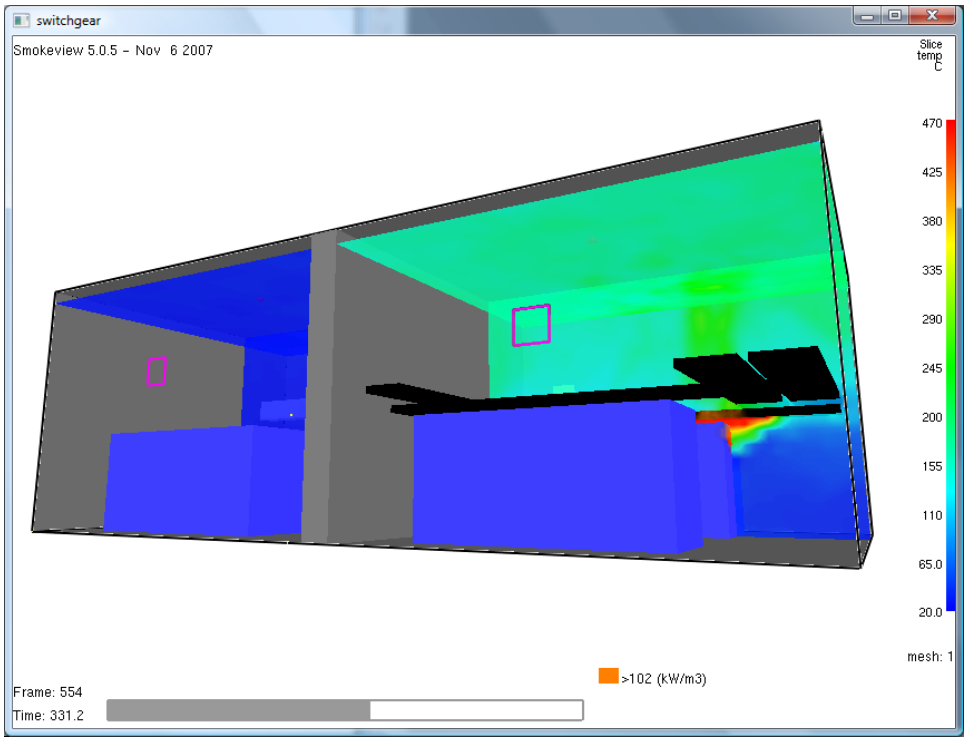

Figure 5.19. Temperature contours on slice planes at 330 seconds

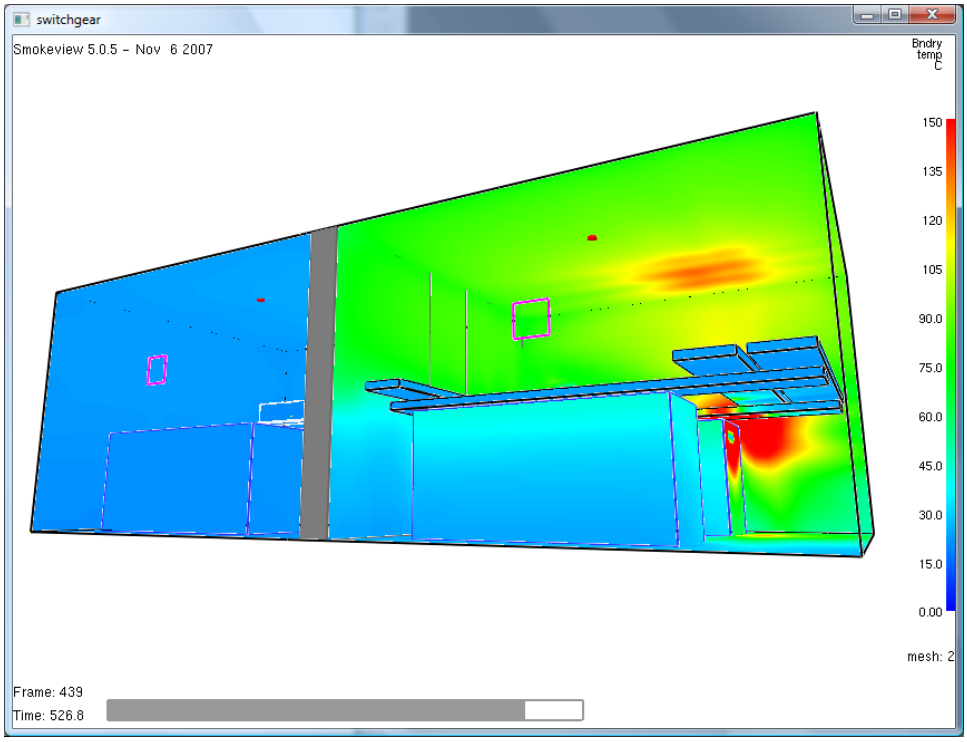

Figure 5.20. Temperature contours on the solid surfaces at 525 seconds

To view time history results

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results...**.

2. A dialog will appear showing the different types of 2D results that are available. Select **switchgear_devc.csv** and click **Open** to view the device output. The first display will be the layer height in Room 3A, Figure 5.21.

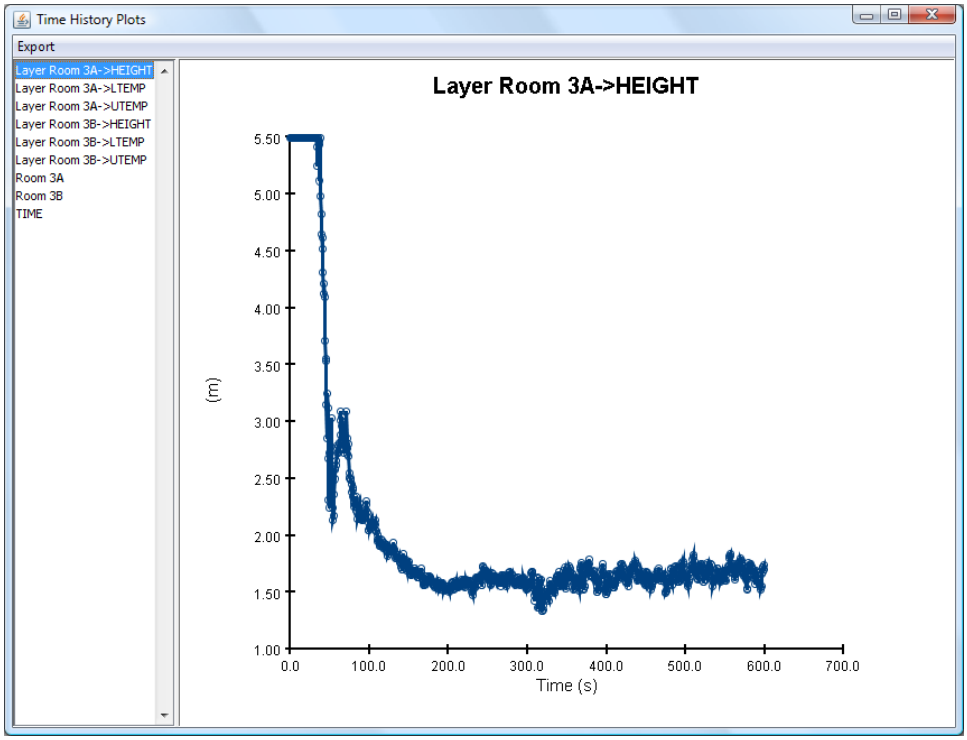

Figure 5.21. Layer height in room 3A

References

[McGrattan, Klein, Hostikka, and Floyd, 2007] Kevin McGrattan, Bryan Klein, Simo Hastikka, and Jason Floyd. *Fire Dynamics Simulator (Version 5) User's Guide*. July, 2007. NIST Building and Fire Research Laboratory. Gaithersburg Maryland USA . LBNL-43134.

[U.S. NRC, 2007] Kevin McGrattan, Bryan Klein, Simo Hastikka, and Jason Floyd. *Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications*. May, 2007. U.S. Nuclear Regulatory Commission, Office of Regulatory Research. Washington DC USA . NUREG-1824, EPRI 1011999.

[Klein, 2007] Bryan Klein. *Switchgear example problem, personal communication.*. 2007. National Institute of Standards and Technology. Gaithersburg MD USA .